NPS-57VA71111A

Naval Postgraduate School

PROCEEDINGS OF THE WORKSHOP ON
FLOW IN TURBOMACHINES

by

M. H. Vavra
K. D. Papailiou
J. R. Woods, Jr.

16 November 1971

Approved for public release; distribution unlimited.

NAVAL POSTGRADUATE SCHOOL
Monterey, California 93940

Rear Admiral A. S. Goodfellow
Superintendent

M. U. Clauser
Provost

Abstract:

This report contains the proceedings of the Workshop on Flow in Turbomachines which was held in December 1970 under the sponsorship of the Naval Postgraduate School and the Naval Air Systems Command. The workshop participants included representatives from the academic community, industry, and government. The participants at the workshop are indicated by an asterisk in the distribution list.

This task was supported by: Naval Air Systems Command, Code 310
AIRTASK No. A310310A/551A/1R01003010 Title (1)

TABLE OF CONTENTS

WELCOME ADDRESS

by

Rear Admiral Raymond J. Schneider, USN
Assistant Commander for Research and Technology
Naval Air Systems Command

The Naval Air Systems Command (NAVAIR) is sponsoring this Workshop on Flow in Turbomachinery as part of its management routine for planning mission-oriented research programs. It is obvious that a commodity-oriented organization such as NAVAIR, which carries the responsibility for aeronautical RDT&E in the Navy, must carefully control its resources in order to obtain an optimum return for the scarce R&D dollars. We can neither afford to support all relevant proposals, nor sponsor research which cannot be expected to result in a direct contribution to our technological mission. Instead the Command must concentrate on those problems which are significant barriers towards achieving its technological and operational goals. Since there are usually many more problems than can be accommodated within a limited budget, skillful selection becomes imperative.

The basis for NAVAIR's R&D planning is, therefore, the identification of key technological problems from the standpoint of the Command's mission, and the proper translation of these problems into well-defined research and development objectives. This task, however, requires all the advice and assistance which can be obtained from the technical community. As a consequence of this management philosophy, NAVAIR fosters strong involvement of the R&D community in all phases of program planning and control. Ample use is made of consultant and ad hoc study groups; experts are invited from the Navy laboratories to join Headquarters for special planning assignments and state-of-the-art reviews, and by sponsoring Workshops exchange of information and ideas is stimulated in technological areas of importance to the Command.

It is the objective of this Workshop to engage outstanding engineers and scientists from the aeronautical industry, from universities and from the Government in technical discussions on a subject which is of prime interest to Naval aviation, namely, how to expand the scientific/technological basis for the design of improved systems for military aircraft. Thus the emphasis in this Workshop will not be on achievements but on possible advancements of the state of the art in airbreathing propulsion. The format of a discussion meeting has been chosen to provide an informal and congenial atmosphere, and invitations have been extended not so much because of the affiliation of the invitee with a particular organization, but rather because of his reputation as an expert.

There couldn't be a better choice for holding this Workshop than under the auspices of the Aeronautical Department of the Naval Postgraduate School, which has a reputation in the Navy both for its research work and its dedication to the technical training of our Navy officers. NAVAIR recognizes the academic excellence of the Postgraduate School in a very practical way by sponsoring a considerable amount of aviation-oriented R&D work at Monterey. The Command is particularly cognizant of the unique research potential and professional reputation of the School's Turbo-Propulsion Laboratory under Professor Vavra and his colleagues. Although the Navy has very capable E&T facilities for engine technology, it is still lacking a true center for turbomachinery research to lead the Navy's R&D efforts in this technological area. Given the proper support, the Turbo-Propulsion Laboratory could well become the nucleus for such a Navy center of excellence. All important requisites are already available: outstanding professional talents, suitable facilities, and last but not least, a conducive academic environment.

In summary, the Naval Air Systems Command is very grateful to the Postgraduate School for being the host of the meeting, and to the many distinguished experts and guests for kindly following the invitation to the Workshop. The Command is certain that the meeting will be helpful for both "producers" and "consumers" by establishing scientific discourse across corporation and agency boundaries, and exposing problem areas for which research could produce the most profitable payoff in terms of an expanded technological base for future Naval aeronautical engine development.

KEYNOTE ADDRESS

by

Rear Admiral Carl O. Holmquist, USN
Chief of Naval Research

Preliminary Remarks.

It gives me a great deal of pleasure to welcome you all to this Workshop on Flows in Turbomachines. Personally, I have been looking forward to this occasion with a great deal of anticipation and interest.

My graduate work, back in the early 1950's, was closely related to some of the problems I know you will be discussing during this symposium, and I am particularly delighted and honored to give the keynote speech and to join in the discussions.

As Chief of Naval Research, I consider workshop meetings of this type extremely valuable in helping to solve one of our greatest problems in research -- the problem of communication and coordination among those individuals conducting research in a particular area of technology. I hope that each of you will take maximum advantage of this opportunity to exchange ideas and information with your colleagues.

This morning, it is not my intention to talk about detailed problems in those areas of technology associated with flow processes in turbomachines. Rather, I would like to discuss trends, broad concepts and major problems that are associated with the attainment of our ultimate goal of improving the performance of military turbomachinery, knowing full well, of course, that any improvements we can make in military power plants will eventually be reflected in and enjoyed by commercial power plants.

We are all interested, I'm sure, in the research and development we should be doing today in order to meet our needs for tomorrow. In this context, then, I would like to discuss briefly where we have been, where we are now and, hopefully, where we are, or should be going. I intend to flavor this discussion with some of my own ideas of where I think we should be headed, and hope that some of these ideas will be sufficiently controversial as to generate discussions and, perhaps, arguments among the participants in this Workshop

The jet engine is not new. As we all know, the basic concepts are quite old. Heron of Alexandria suggested a device that today would be called a gas turbine in about 130 B.C. The first patent for a gas turbine operating on a cycle quite like present-day turbojets was issued to John Barber of England in 1791, almost two hundred years ago. The first turbojet engines producing useful work were developed almost simultaneously by the Germans and the British during World War II.

One might ask the question as to why it took so long (almost 2,000 years) to develop a turbojet engine that would produce useful work. In retrospect, the answer is obvious. The science of aerodynamics and fluid flows had to progress to the point where we could design efficient airfoils for blade shapes before we could produce a workable turbojet engine. In addition, of course,we had to develop materials and alloys that could withstand the high temperatures and critical stresses involved in turbomachinery. We had also to learn to design combustors and other critical components for the jet engine.

In the past 30 years there have been almost steady improvements in turbojet engines -- better materials, improved internal aerodynamics, higher stage loadings, component improvements and the increased use of variable geometry in various parts of the engine.

In the late 1940's we operated turbojets in the speed range from 300 to 500 miles per hour with thrust-to-weight ratios from 2 to 3. Steady improvements were made until about 1960 when there was a noticeable slow-down in both aeronautics and turbomachinery research and development as our national emphasis was shifted to space research and development.

However, during the last five years, there appears to have been a renewed effort in government programs supporting air breathing propulsion systems. There are now many very active programs underway in inlet, compressor, combustor, turbine, nozzle and component technology.

In recent years, as we have increased pressure ratios and peak cycle temperatures, made larger and larger engines, increased the bypass ratio and introduced supersonic blading,the fluid flow problems associated with turbomachinery have become much more complex.

Of particular concern to this Workshop is the increased emphasis now being given to aircraft -- propulsion system interactions, the effects of inlet flow distortions on engine operation, and the internal flow field through all sections of the turbojet engine.

<u>Figure 1</u>. This Figure lists 18 variable engine parameters which are all interdependent and in need of precise control if we are to have a turbojet engine which will operate efficiently over a wide flight envelope <u>off design point</u> and under all flight conditions, such as:

Acceleration and Deceleration
Ascent or Descent
Various Angles of Attach or Yaw
Atmospheric Variation (such as clear air turbulence)
Hot Air Ingestion (Rocket Firing)

Each of these considerations involve a <u>matching problem</u>. In toto they represent a severe challenge to this workshop and to the entire technical community involved in the design and development of improved turbomachinery.

Let us now examine some of the important airplane and engine parameters and the projection of these parameters into the future. Such projections, of course, should give us an indication of the direction we are going and what we might hope to accomplish in the future. In the last analysis, however, military requirements will determine, as they always have, the precise direction of our efforts.

Figure 2. This figure shows a plot of the maximum speed of jet fighters versus the year of first flight. This plot appeared in a paper presented by Mr. Joslin of Pratt-Whitney at the recent Propulsion Joint Specialists Conference in San Diego.

The plot clearly shows that our efforts in the early 1950's were to increase the maximum speed of jet fighters. Since about 1960, however, the curve has flattened out. With the exception of the YF-12, which has never gone into production, we have not built an operational fighter with a maximum speed of more than about Mach 2.3.

In spite of all rumors to the contrary, it is my personal feeling that we probably never will, although the technology to accomplish this has been available for some time. I do not even believe that we have a fighter requirement for sustained flight at any supersonic speed. The basic reason is the guided missile, which has become a standard air-to-air armament for our fighters.

For fighters, we need lots of agility and maneuverability, and, at times, short bursts at supersonic speeds to get into a firing position; but even the wildest dreamers have never proposed dogfights at Mach 3.

This is not to say that there are no requirements for turbojet engines which can operate at speeds in excess of Mach 2.5. There obviously are. The supersonic transport is one. There may also be requirements for supersonic reconnaissance aircraft that operate at speeds of Mach 3 or higher. In addition, some missile applications may require turbojets (or other air-breathing power plants) for propulsion at speeds that may well exceed Mach 2.5.

Figure 3. As far as transport aircraft are concerned, there has been a strong trend toward larger and larger aircraft and engines as can be seen on this figure. (C-5A, Boeing 747, SST).

From the technical viewpoint, these changes have been possible because of better materials, improved component performance, better cooling techniques and improved analysis techniques for structures and fluid flows.

From the economic viewpoint, large commercial transports reduce operating costs per ton-mile or per passenger-mile. From the practical viewpoint, fewer but larger transport aircraft will help to reduce the congestion in our airways.

The figure shows that it should be possible to increase the size of commercial transport aircraft to about 1,500,000 pounds by 1990. Other people have predicted that the total weight of transport aircraft could reach 5 million pounds. I'm sure that this is technically feasible; however, from an economic standpoint it may not be as attractive, since most of the country's airport runways, taxiways and parking areas would have to be rebuilt to withstand the increased loads -- a multi-billion dollar construction program.

Figure 4 gives predictions by industry of turbojet engine growth possibilities. It shows that turbofan engines in the 60,000-pound-thrust class could evolve to power a 900,000-pound transport in the 1980 time period, and that engines approaching 100,000 pounds of thrust could be built in the 1995 time period. I would hasten to add that this slide shows what appears to be technically feasible and may bear no relation to what we actually will do in a practical sense.

I would now like to show the trends over the past few years in some of the important engine performance parameters.

Figure 5 (From Pratt-Whitney Data) illustrates the trends in thrust-to-weight ratio since 1945 for turbojet and turbofan engines. In the past 15 years, the development of titanium technology and constant improvement in fabrication techniques, high temperature alloys, turbine cooling and components have approximately doubled the thrust output per pound of engine weight.

The development of fiber and composite materials, high temperature alloys, new blade cooling techniques and continued improvement in internal aerodynamics -- all promise further improvement in jet engine thrust-to-weight ratios in the future.

As you can see, the improvement trend is almost linear. The advanced technology engine now under development for the Navy and the Air Force should have a T/W value even higher than those shown for the 1970's.

Extrapolating the curve, we can easily predict T/W ratios of 10 to 12 in the next 10 years. The realization, some day, of stoichiometric engine should give even higher values.

Figure 6 (Also from P&W data) shows the trend of thrust specific fuel consumption for turbojets and turbofans since 1945. Since 1950 there had been a reduction of about 50% in TSFC. Within sight are TSFC values as low as .55 (lb/hr per pound of thrust).

Figure 7 shows TSFC as a function of bypass ratio for various overall pressure ratios and turbine inlet temperatures. There has also been a similar improvement trend in engine thrust per unit airflow weight over the past twenty years. This improvement has resulted from increases in turbine inlet temperatures and from cycle variations.

I would now like to show some of the progress over the years in improving component performance.

Figure 8 shows the trend in compressor pressure ratio and increased compressor unit temperature over the past 20 years. Compressor pressure ratios have increased from about 4 to nearly 25. It is predicted that, within the next 10 years, pressure ratios will go to 40 at compressor exit temperatures of about 1200°F.

Efficiencies of both turbines and compressors have increased from about 82% to about 90% in the past 20 years. Only modest improvements can be expected in the future in these efficiencies.

Compressor improvements have resulted primarily from improved blade design, higher tip speeds, higher pressure ratios and better stage matching. Turbine improvements have resulted from increased turbine inlet temperatures, improved work loading, and (as in the case of the compressor) improved blade design.

Figure 9 shows one of the most spectacular component improvements in the past 20 years -- the increase in turbine inlet temperature, currently, we are at the 2400 - 2500° F level and are working toward the 3000° F level.

The large increases in allowable turbine inlet temperature have been made possible by various blade cooling techniques. These temperature increases began in about 1960 at about the same time that significant increases in compressor pressure ratios were realized.

Since the working level of today's turbine materials is limited to about 1900° F (with melting temperatures starting at about 2300°F), more sophisticated and more effective ways to cool turbine blades are needed if we are to operate at higher and higher turbine inlet temperatures.

Figure 10 shows turbine-blade-cooling trends over the years and shows that we will need transpiration-cooled blades (or some other new and better scheme) in order to significantly increase turbine inlet temperatures in future years.

Recently, there have been engine studies and configuration plans for a number of engines with turbine inlet temperatures in the range from 2600° F to 3000° F.

Beyond this, we see the possibility of the so-called "ultimate" in jet engines -- the stoichiometric engine. This engine would have to use advanced materials and blade cooling techniques in order to successfully operate at such turbine inlet temperatures.

Figure 11 shows engine inlet temperatures, compressor temperature rise and the variation of turbine inlet temperature with Mach number for the stoichiometric engine.

The chart shows the theoretical limits within which the turbine engine must operate. The inlet temperature is determined by the flight Mach number. There is, of course a rise in total temperature through the compressor. The heating value of the fuel, burned at the stoichiometric ratio, determines the maximum turbine inlet temperature possible. The overall efficiency of the engine is largely determined by how close we can operate the top line.

I would like to conclude this discussion by talking about some of the major problems that I see facing the research and development community in the design and development of improved turbomachinery for the future.

Figure 12 is a list of items which I believe are the major problem areas we face. The list is by no means comprehensive or all-inclusive.

(1) Stoichiometric Engine.

I believe that the ultimate goal of almost all of our research and development efforts in turbomachinery should be the development and final production for operational use of the stoichiometric engine.

This goal may never be completely realized. However, any improvements we can make in compressors, turbines, combustors or components toward this goal will certainly improve the future jet engines we build.

The realization of a stoichiometric engine should give us a significant improvement in engine thrust-to-weight ratio and in specific fuel consumption. It would allow us to build fighter aircraft of significantly improved performance. Without an improved engine it is literally impossible to build an improved aircraft for the same mission. Under contract with the Air Systems Command, the Allison Division of General Motors continues a successful development program of this type propulsion system.

In addition to combustor design problems and blade cooling problems with a high temperature engine, we also have the serious problem of increased radiation of heat to other parts of the engine and the effect of this heat on structures and components. Much development effort will be required before these problems are solved.

(2) Flow Analysis in Large Compressors and Turbines.

Although we have analytical procedures which are fairly useful in the analysis of two-dimensional, compressible flow and in simplified three-dimensional cases, to my knowledge we have no analytical method for handling three-dimensional, compressible flow in large compressors and turbines. We need a great deal of theoretical work in this area.

In the area of non-uniform flows and non-steady flows we know even less, and much theoretical and experimental work needs to be done so that we can better predict flow patterns over stator and rotor blading under these conditions.

The Russians apparently design and build compressor and turbine blading that differs somewhat from ours in that unique aerodynamic and contstruction techniques are involved. Such designs should be studied to determine general applicability and pinpoint important variations.

(3) Materials and Fabrication Techniques.

We need to continue a strong effort in that never-ending search for new and improved materials for blades, casings, bearings and other components of our engines. We also need improved and less expensive fabrication techniques.

With the new technology of composite materials and new filament materials such as glass, carbon and boron fibers, we should be able to fabricate lighter compressors with thinner blades of equal stiffness and strength.

We should (possibly) be able to fabricate more sophisticated contours than presently used and (also possibly) solve some of the materials problems associated with the supersonic compressor.

(4) Blade Tip Losses.

We need to reduce blade tip losses in all of our designs and improve tip seals. More satisfactory solutions to this problem could improve compressor and turbine efficiency significantly. Of particular concern is the blade seal problem under conditions of heavy water and ice injestion in the engine.

(5) Inlet Design.

One of our most important and ever-present problems is that of distorted and unsteady flow in engine inlets, particularly at high angles of attack and yaw.

We have had so much trouble in the past with side inlets with boundary layer bleed (such as the F-111 and F-4) that in our new designs we have used essentially two-dimensional inlets which are physically removed from the fuselage and, hence, not affected by fuselage interference flows and distortions at high angles of attack and yaw.

The F-14 has inlets out in the free stream. The new S-3 ASW carrier airplane also has a simple circular inlet placed far enough out on the wing to preclude inlet pressure distortions from fuselage interference patterns.

(6) Reliability and Maintainability.

This talk would not be complete without mention of our ever-present problem of the "abilities" as applied to turbomachinery. We still have drastic problems of reliability and maintainability with every new model of jet engine we introduce.

These problems usually stem from very simple design problems -- improper heat treatment of parts, quality control, defective welds, the wrong radius on a fillet, a defective lock washer, the wrong bolt size, etc. Our experience shows that over a long, agonizing period of time, the manufacturer usually is able to correct most of these defects.

With all of our experience to date, it is incomprehensible that we should still have such simple (but disastrous) problems in new designs. We still need a great deal of effort in this area of reliability and maintainability.

(7) "Sheet-Metal" Engine.

For certain missile applications it appears as though a "poor-man's" version of the turbojet engine is the best propulsion plant. Such an engine would not have to be built for long life and high reliability (life of 1-10 hrs ?).

With our present knowledge of turbojets, we should be able to make inexpensive "sheet-metal" designs that can be mass produced in quantity for this type of application. Such a design should be a real test of the ingenuity of the designer and the manufacturer.

(8) "Re-engineered" Engines.

Our turbojets are getting far too complicated and expensive. The price of our advanced technology engine is about three-quarters of a million dollars.

I would like to propose that we should assemble a group of the country's leading engine designers to completely re-engineer a new engine design after it has demonstrated its design performance.

Such a re-design would involve every part in the engine and the manufacturing processes used to make each part. The goal of this re-design should be to maintain the same performance but to simplify, improve reliability and maintainability, and reduce the cost of manufacturing wherever possible.

Re-engineering would cost money, of course, but in the long run it should result in a less expensive, simpler and more reliable engine.

(9) Internal Instrumentation.

With increased engine temperatures and larger engines with more complex three-dimensional flows, we need improved instrumentation for monitoring pressures and temperatures throughout the engine -- particularly for tests of experimental designs, cascade tests, etc.

(10) Infra-red Signature.

A major consideration in every engine design should be the IR emmission because of the problem of IR-seeking missiles of the Sidewinder type. We need new and ingenious schemes for reducing the IR signature of operating turbojets. As the turbine inlet temperatures continue to rise, this problem will become more and more serious.

(11) Noise.

The inlet and exhaust noise from a jet engine is of little concern to the military, since we are not generally willing to sacrifice performance or increase cost in order to reduce noise.

However, in commercial applications this is a major consideration. We need theoretical and experimental work which will give us a better understanding of the unsteady time-dependent flows in compressors and fans and a better knowledge of the mechanism of noise production in jet exhausts.

(12) Pollution.

As in the case of noise the military is not generally willing to sacrifice performance or increase cost to reduce pollution from jet engines. However we have a strong interest in reducing the smoke emission from jet engines to reduce the possibility of visual detection of our aircraft by the enemy.

We need additional work which will lead to a clearer theoretical explanation of the formation of carbon and NO compounds in a combustor. We also need continued improvement in combustor design to eliminate the smoke trail of turbojets and turbofan engines for both military and civilian applications.

Closing Remarks.

The next group of papers to be presented today were planned to cover state-of-the-art summaries pertaining to our analytical capabilities in handling various flow processes in turbomachines. This type of information, once digested and considered in the light of the possible performance projections we visualize, should prove beneficial to future engine development. We can honestly say there is an important contribution that can be made by examining the needs for research pertinent to flow processes in turbine engines. Further, there is a definite need to better coordinate our various efforts to the end that maximum progress is made consistent with the limited funding that we have. The subjects covered by todays' discussions are important and represent areas in which we feel we probably should devote somewhat more attention than we have in the past. Our stewardship, as managers of research, with some of us intimately involved with selection of projects, naturally dictates that we concern ourselves with the details of future efforts. It is in this context that I feel the time is opportune to proceed with this Workshop meeting.

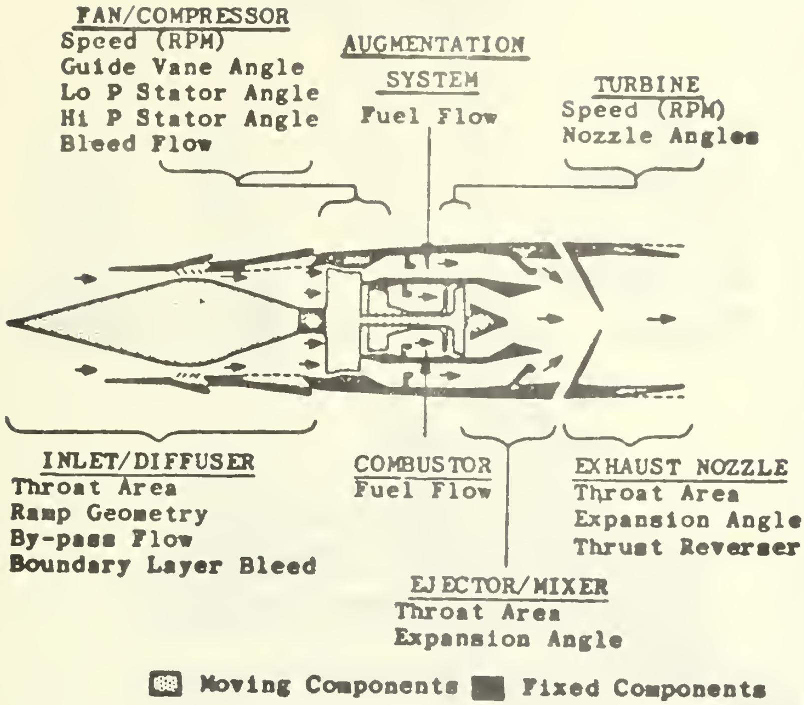


FIGURE 1. Advanced Turbine Engine Variable Parameters

Source: ICAS Paper No. 70-45

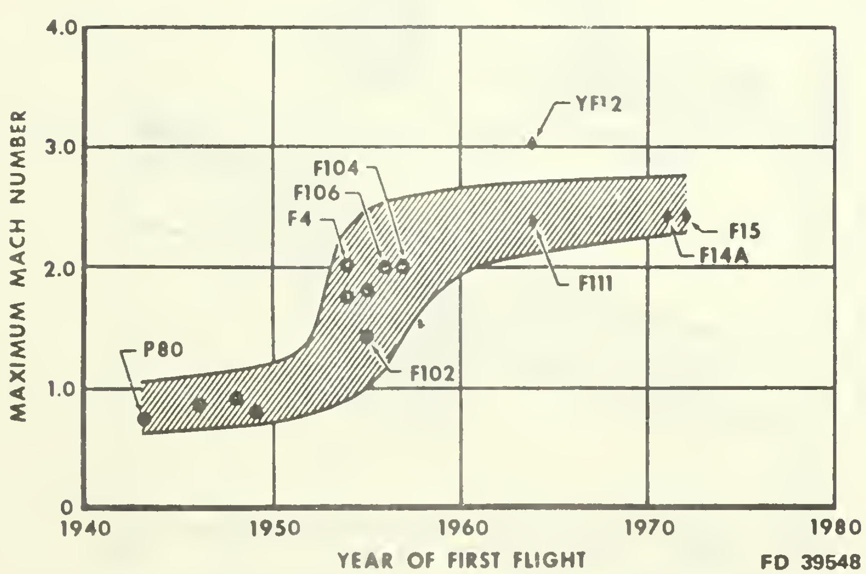


FIGURE 2. Maximum Speed of Jet Fighters

Source: AIAA Paper No. 70-943

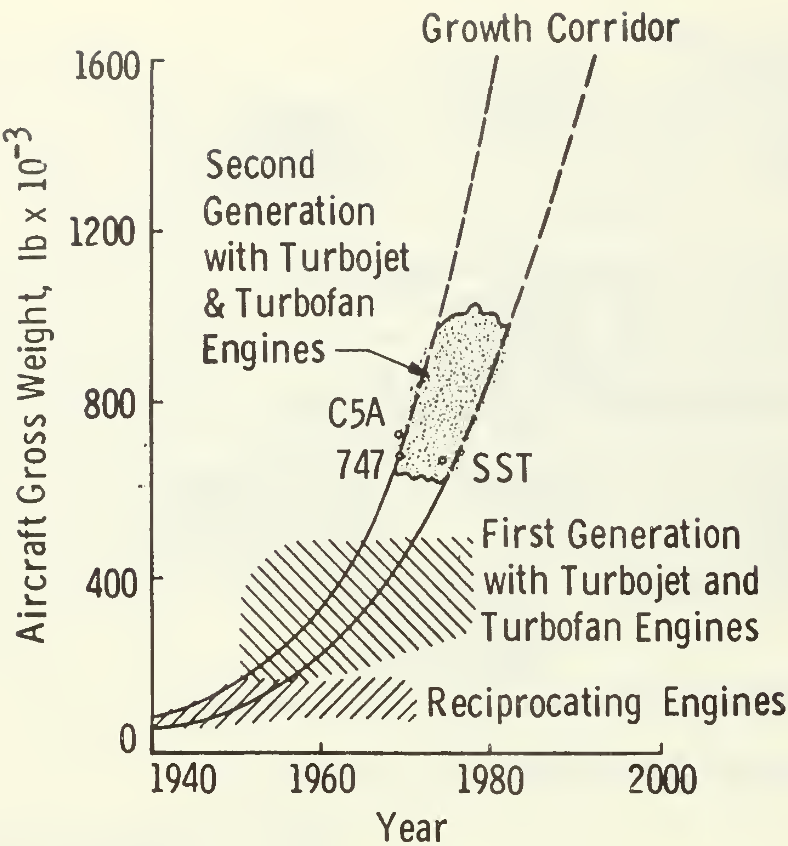


FIGURE 3. Aircraft Growth Trends
Source: ICAS Paper No. 70-45

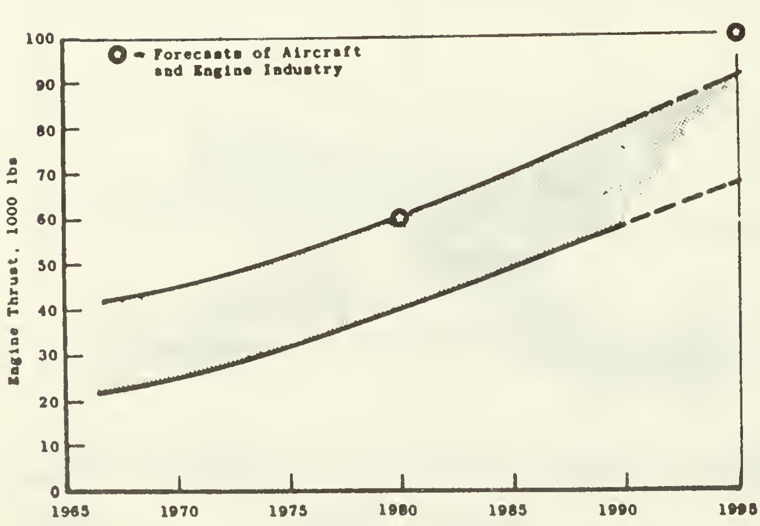
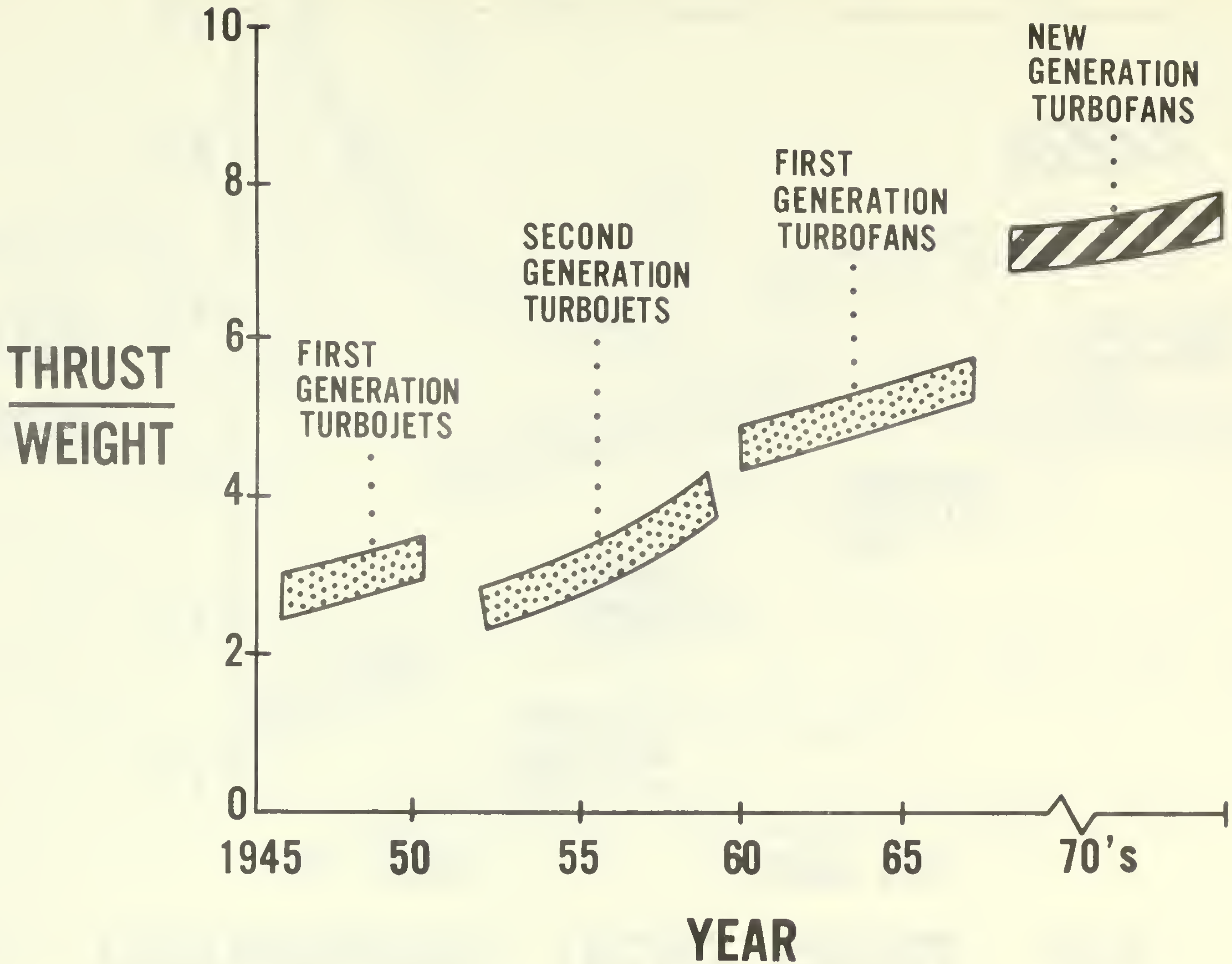


FIGURE 4. Engine Growth Predictions
Source: ICAS Paper No. 70-45

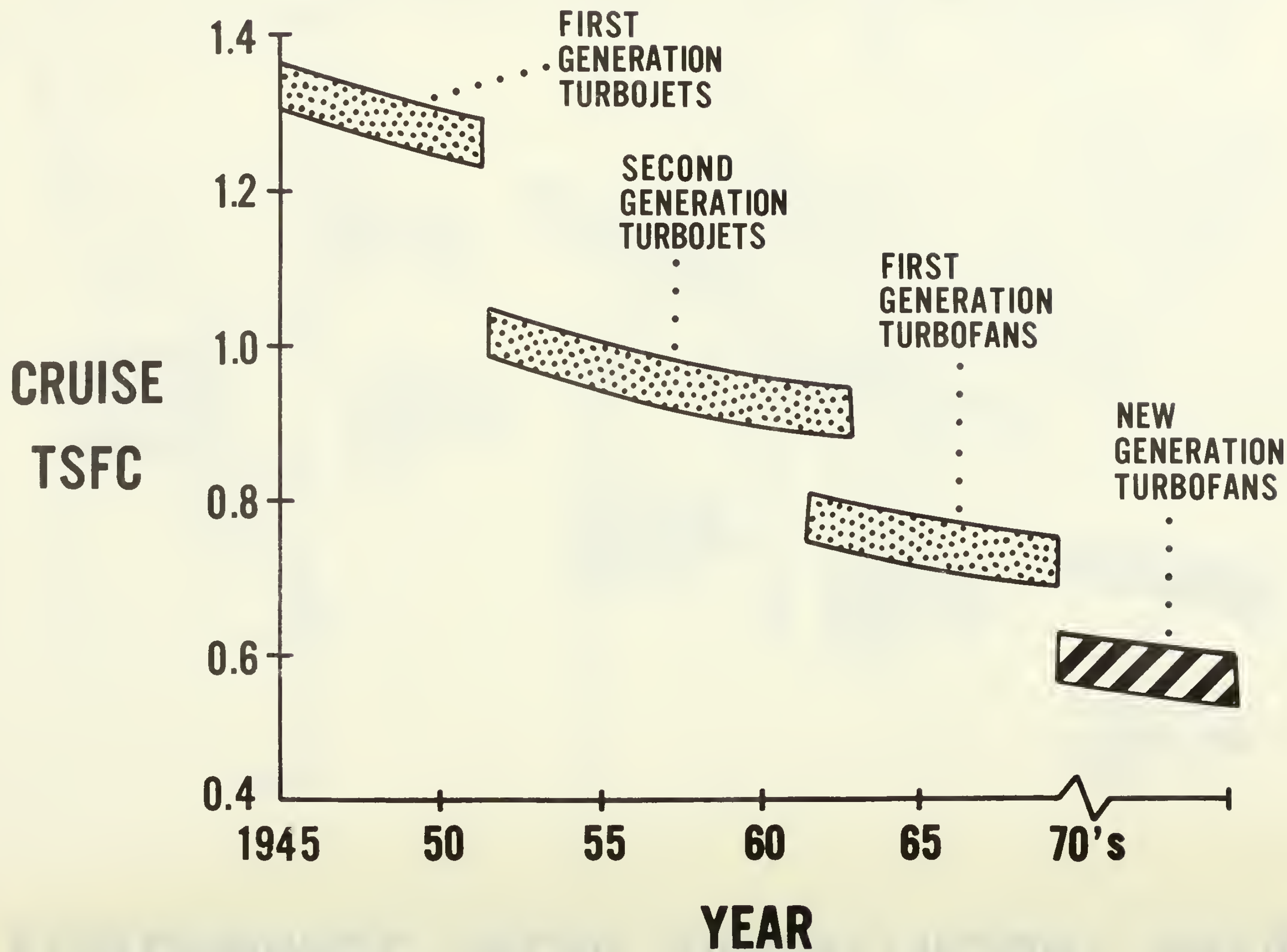
FIGURE 5. PERFORMANCE TREND THRUST/WEIGHT RATIO



M-45213
R670507

FIGURE 6. PERFORMANCE TREND CRUISE TSFC

0.8 MACH 36,089 FEET



M-45214
R670507

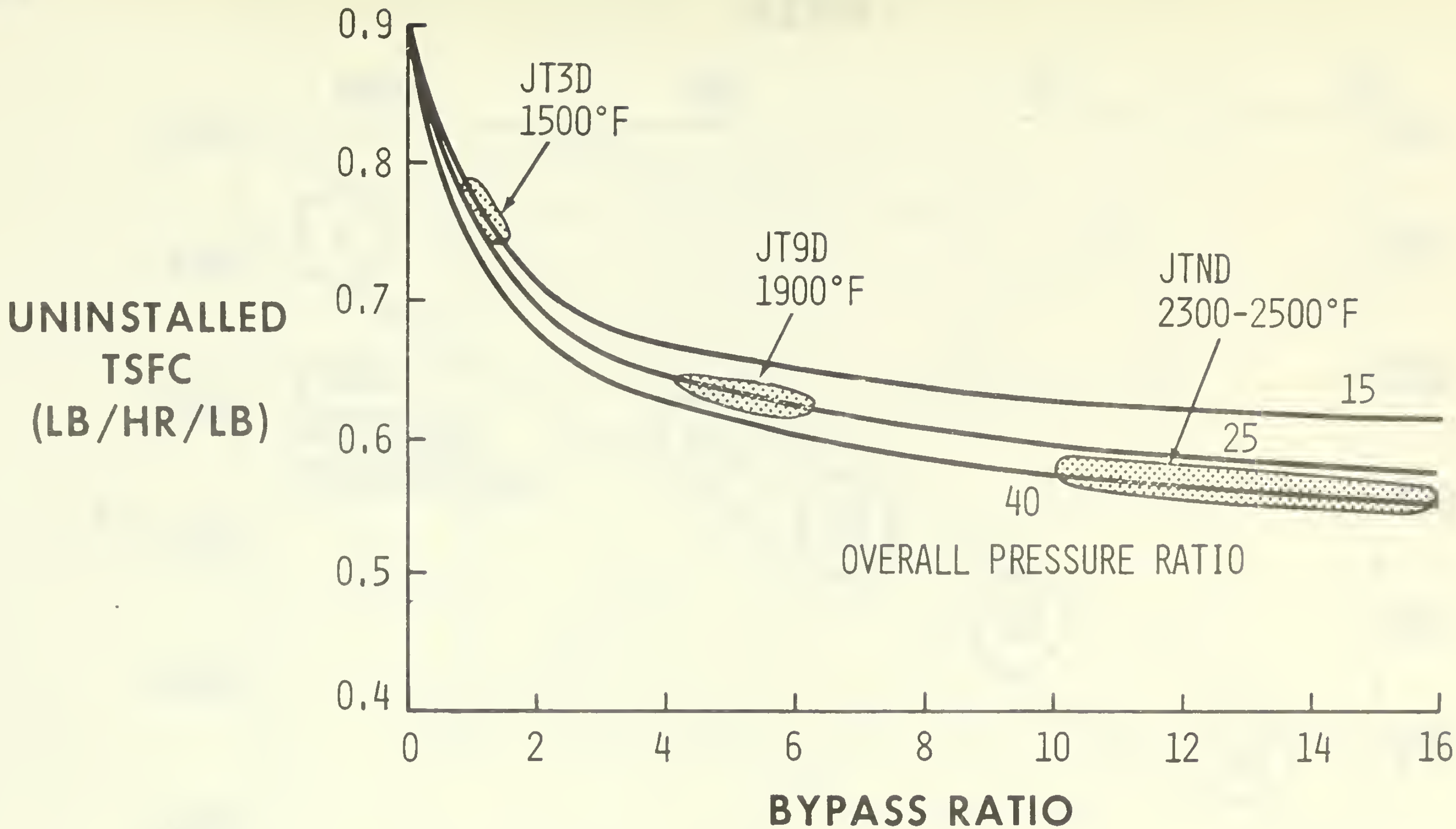
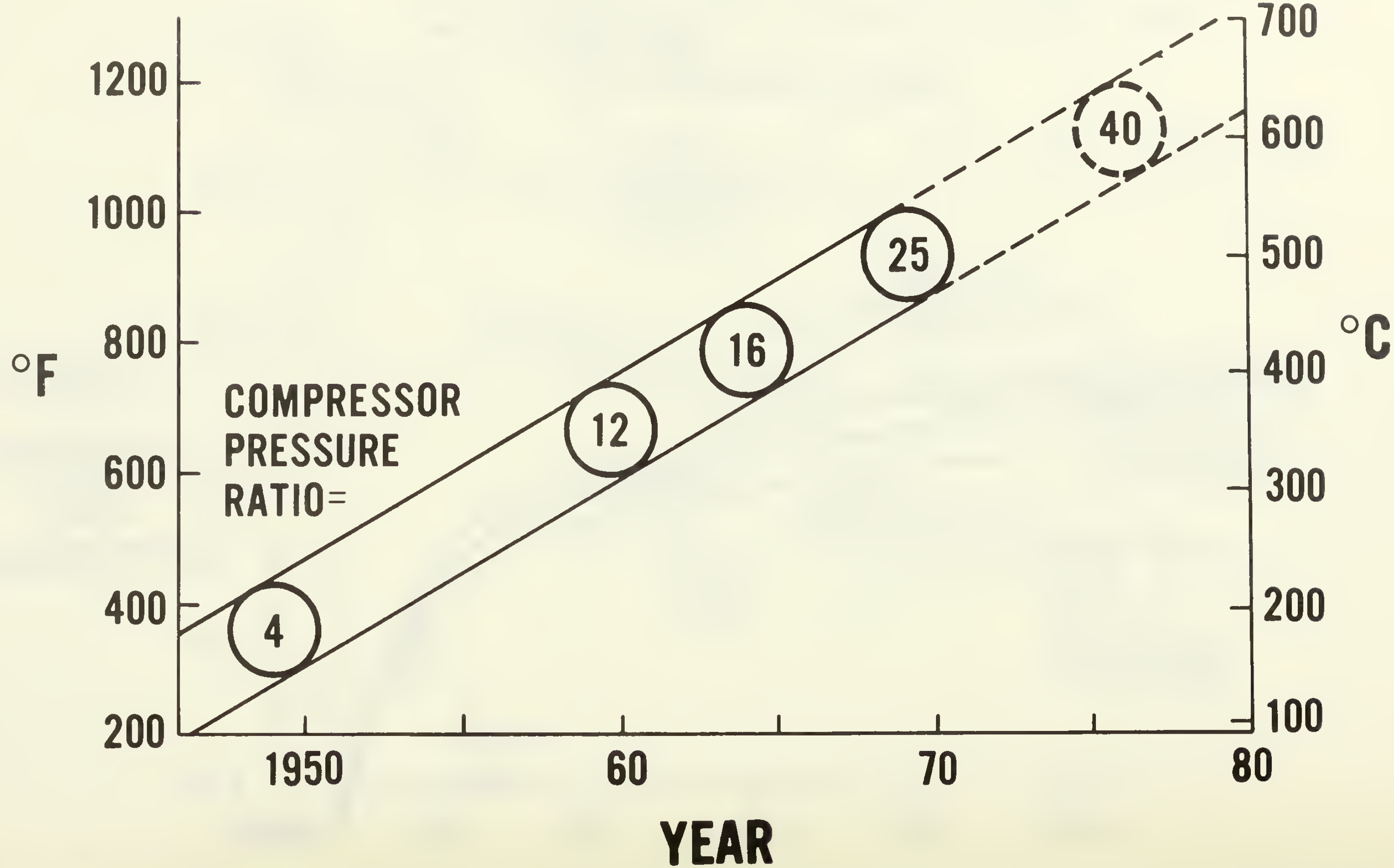


FIGURE 7. IMPROVEMENT POSSIBLE IN INSTALLED SPECIFIC FUEL CONSUMPTION (MACH 0.9, NOZZLE C_V = 0.99)

FIGURE 8. **ENGINE CYCLE TRENDS**

COMPRESSOR EXIT TEMPERATURE

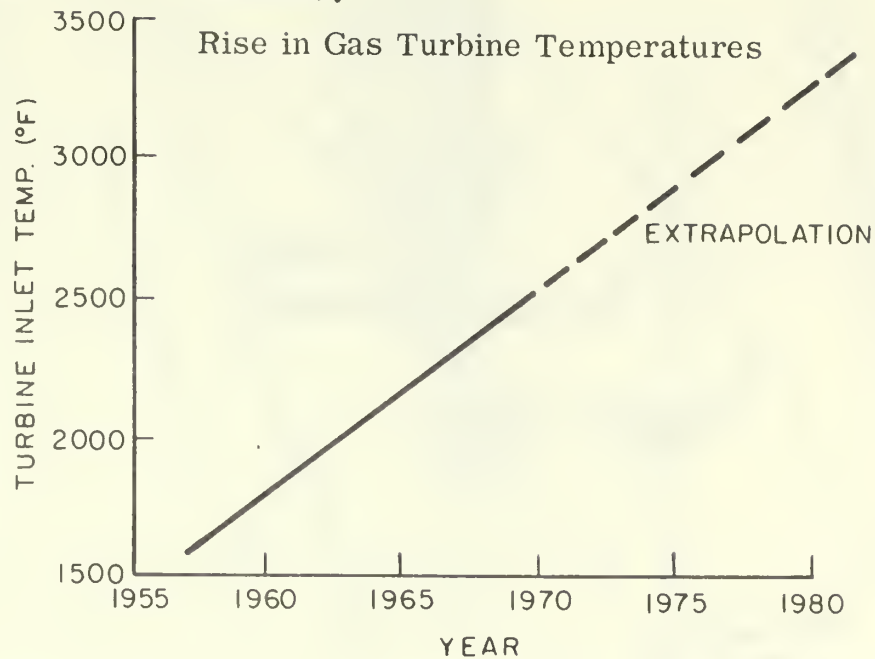


Pratt & Whitney Aircraft DIVISION OF UNITED AIRCRAFT CORPORATION

Ti M-54009
681308

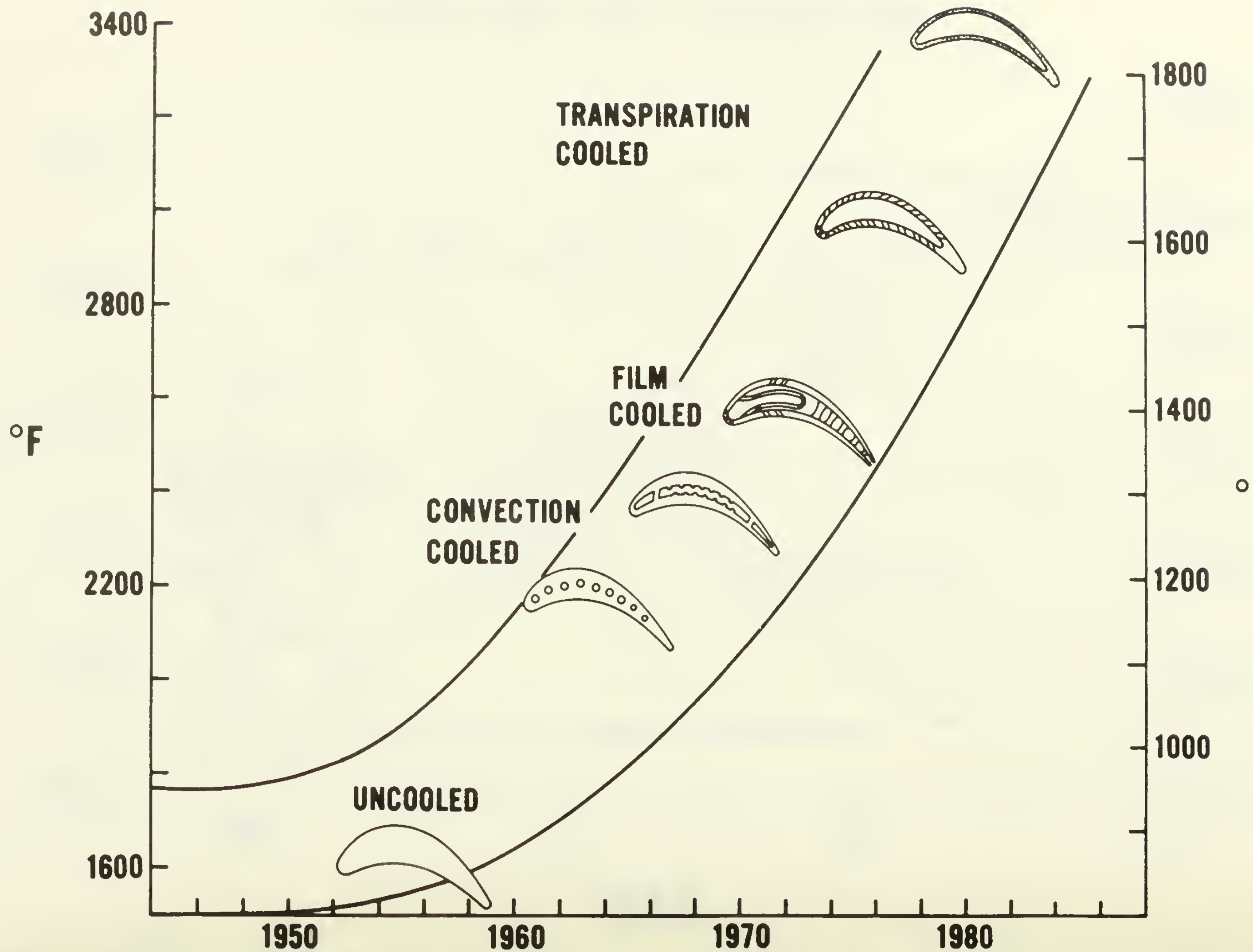
FIGURE 9.

Rise in Gas Turbine Temperatures



Source: ONR Report DR-156

FIGURE 10. TURBINE BLADE COOLING



Pratt & Whitney Aircraft DIVISION OF UNITED AIRCRAFT CORPORATION

J1225-1
R702808

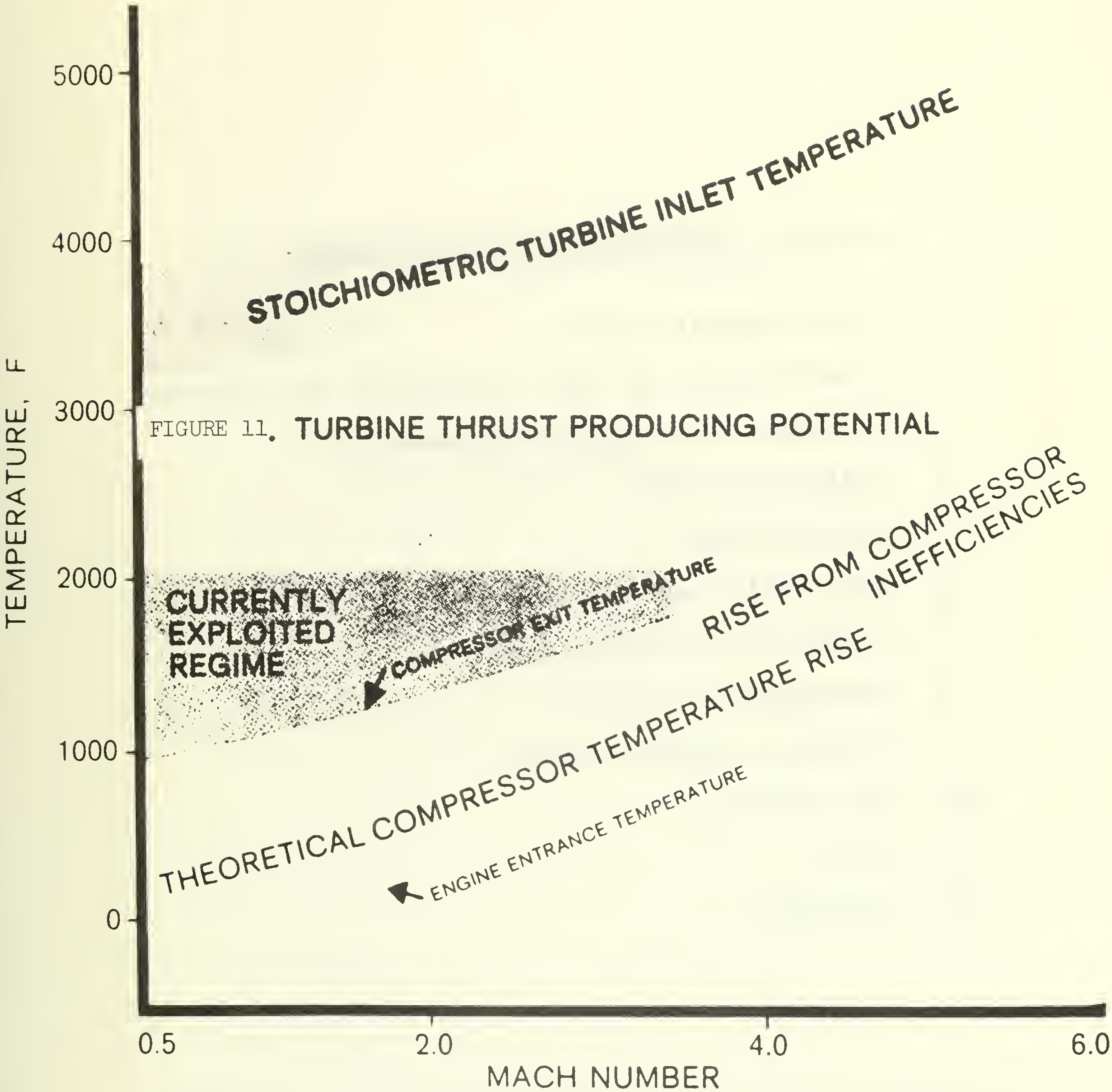


FIGURE 11. TURBINE THRUST PRODUCING POTENTIAL

Source: Astronautics & Aeronautics
July 1969

FIGURE 12. TURBOMACHINERY PROBLEM AREAS

1. STOICHIOMETRIC ENGINE
2. FLOW ANALYSIS IN LARGE COMPRESSORS AND TURBINES
3. MATERIALS AND FABRICATION TECHNIQUES
4. BLADE TIP LOSSES
5. INLET DESIGN
6. RELIABILITY AND MAINTAINABILITY
7. "SHEET METAL" TURBOJET
8. RE-ENGINEERED ENGINES
9. INTERNAL INSTRUMENTATION
10. IR SIGNATURE
11. NOISE
12. POLLUTION

NON-STEADY PHENOMENA IN TRANSONIC AND SUPERSONIC FLOWS AND POSSIBLE METHODS OF SOLUTION

by

Dr. M. F. Platzer
Associate Professor
Naval Postgraduate School

In my presentation today I would like to cover only some of the basic aspects of unsteady,transonic flows and discuss some of the main methods that have been developed to solve the unsteady,transonic small-perturbation equation.

Figure 1 shows the familiar wave patterns that are produced by a disturbance source that travels in a compressible medium at either subsonic or supersonic speeds. Figure 2 presents the case of a source in a transonic freestream. It is seen that parts of the wave fronts, referred to as "receding waves", move upstream very slowly relative to the source, whereas others, the "advancing waves", move downstream with a velocity approximately equal to twice the speed of sound. Assuming now a continuous spatial distribution of disturbance sources, it is readily understood from the bottom figure of Figure 2 that the receding waves will tend to cancel provided the wavelength is small compared to the body reference length, i.e., provided the frequency is high enough. However, if the sources oscillate very slowly, pressure waves of the same sign will have sufficient time to interact and thus may eventually build up to higher amplitude waves and to shock waves. From this brief physical discussion we can draw the conclusion that sufficiently unsteady transonic flows can be analyzed by linearized theory, whereas only mildly unsteady transonic flows or steady transonic flows will have to be based on a nonlinear equation. A rigorous investigation shows that the condition for linearization may be stated as

$$k >> |1-M_L| \text{ where } |1-M_L|$$

is the largest deviation of the local Mach number from unity and k is the reduced frequency.

Detailed studies of acoustic wave propagation in non-uniform transonic flows were performed at the National Aerospace Laboratory in Amsterdam [1]. Figure 3 shows the position of a wave after equal time intervals which originated from a source located at the three-quarter chord point of the airfoil. The non-uniform flow field is generated by the presence of the airfoil,and the various wave front positions are shown in the upper half-plane. However, if one makes the assumption that the actual non-uniform stationary flow field is replaced by a uniform flow field of, say, a Mach number of 0.8,then the wave positions shown in the lower half-plane are obtained. It is seen that the assumption of uniform stationary flow causes significant differences in time lags when compared with the propagation in the actual non-uniform flow. Unfortunately, our present analysis techniques are mostly based on the uniform flow assumption and thus may introduce appreciable errors. This is even more pronounced at transonic free-stream Mach numbers as shown in the right hand figure for a Mach number of 0.875. The stationary, non-uniform flow field now contains an embedded locally supersonic flow region. The wave pulse sent out by the source at the three-quarter chord point is able to "go around" the supersonic flow region and to enter it from above. The time lags thus introduced are seen to be much larger than those that would be obtained by assuming a uniform flow of Mach number 0.875.

On the basis of frictionless, irrotational flow, we have as the fundamental governing equation the nonlinear potential equation, i.e.

$$\left(1 - \frac{\Phi_x^2}{a^2}\right)\Phi_{xx} + \left(1 - \frac{\Phi_y^2}{a^2}\right)\Phi_{yy} + \left(1 - \frac{\Phi_z^2}{a^2}\right)\Phi_{zz} - \frac{2\Phi_x\Phi_y}{a^2}\Phi_{xy} - \frac{2\Phi_x\Phi_z}{a^2}\Phi_{xz}$$

$$- \frac{2\Phi_y\Phi_z}{a^2}\Phi_{yz} - \frac{2\Phi_x}{a^2}\Phi_{xt} - \frac{2\Phi_y}{a^2}\Phi_{t} - \frac{2\Phi_y}{a^2}\Phi_{yt} - \frac{2\Phi_z}{a^2}\Phi_{zt} - \frac{1}{a^2}\Phi_{tt} = 0$$

where $\Phi(x, y, z, t)$ is the velocity potential function and a is the velocity of sound. As is well known, a complete linearization of this equation is possible for subsonic and supersonic flows in case of small disturbances. As first shown by Lin, Reissner and Tsien [2] a similar linearization at transonic speeds is permissible only if the flow process is sufficiently unsteady. For only mildly unsteady transonic flows, on the other hand, one nonlinear term must be retained, thus, giving the transonic small-perturbation equation

$$[1 - M_\infty^2 - M_\infty^2 (\gamma + 1)\,\Phi_x]\,\Phi_{xx} + \Phi_{yy} + \Phi_{zz} - 2M_\infty^2\,\Phi_{xt} - M_\infty^2\,\Phi_{tt} = 0$$

It is important to emphasize, however, that even in supersonic flows the widely used linearized theory is not uniformly valid. This is best shown by considering two-dimensional, steady supersonic flow past a profile. Figure 4 shows the exact Prandtl-Busemann characteristics solution. The disturbances caused by the profile propagate along the Mach lines, which diverge with increasing distance from the profile. The classical linearized solution (Ackeret theory), on the other hand, predicts Mach lines which are parallel to each other and which coincide with the Mach lines of the undisturbed free-stream. Furthermore, Ackeret's theory predicts no disturbances between the actual shocks originating from the leading and trailing edges and the dashed lines shown in Figure 4. Hence, although Ackeret's theory predicts the pressures on the profile surface quite well, it deteriorates with increasing distance from the profile and fails completely to predict the correct far-field behavior. Clearly, small errors in Mach line inclination can accumulate to large errors along the Mach lines. The basic shortcoming of the conventional linearized theory therefore lies in the a priori fixing of the Mach line inclinations, which is caused by the constant coefficients in the linearized gasdynamic equation.

To overcome this deficiency of the conventional small disturbance theory, Oswatitsch [3] has developed a new small-perturbation theory which introduces the Mach elements as independent variables. In this new theory, therefore, the coordinates and time as well as the velocity components and pressure are the dependent variables. The Mach number, the velocity components and the coordinates may then be expanded as

$$M = M_0 + M_1 + M_2 + \dots$$

$$u = u_0 + u_1 + u_2 + \dots$$

$$v = v_1 + v_2 + \dots$$

$$w = w_1 + w_2 + \dots$$

$$x = x_0 + x_1 + x_2 + \dots$$

$$y = y_0 + y_1 + y_2 + \dots$$

$$z = z_0 + z_1 + z_2 + \dots$$

where the subscript zero refers to the undisturbed flow and the subscripts 1 and 2, etc. denote terms of first and second order in the perturbation parameter, such as the thickness-ratio, flow inclination angle, etc.

Assuming, for simplicity, again steady supersonic flow, one obtains for the Mach lines of an undisturbed uniform flow

$$\xi = x_0 - z_0 \cot \alpha_0 \qquad \eta = x_0 + z_0 \cot \alpha_0$$

where α_0 is the Mach angle of the undisturbed flow. For the inclination of the disturbed Mach lines one has the well-known equations

$$z_\eta = \tan(\alpha + \theta) \cdot x_\eta \qquad z_\xi = \tan(-\alpha + \theta) \cdot x_\xi$$

Expanding these equations with respect to small disturbances of the Mach angle α and the flow angle θ, one obtains the following first order equations

$$x_{1\eta} - z_{1\eta} \cot \alpha_0 = - \frac{M_0}{2C_0} [w_1 \tan \alpha_0 - u_1 k]$$

$$x_{1\xi} + z_{1\xi} \cot \alpha_0 = \frac{M_0}{2C_0} [w_1 \tan \alpha_0 + u_1 k]$$

$$k = \frac{1}{2} [(\gamma + 1) \tan^2 \alpha_0 + (\gamma - 1)] \qquad \gamma = \text{ratio of specific heats}$$

These two equations are easily integrated, thus giving the disturbance coordinates x_1 and z_1

$$x_1 - z_1 \cot \alpha_0 = - \frac{M_0}{2C_0} \int^{\eta} [w_1 \tan \alpha_0 - u_1 k] \, d\bar{\eta}$$

$$x_1 + z_1 \cot \alpha_0 = \frac{M_0}{2C_0} \int^{\xi} [w_1 \tan \alpha_0 + u_1 k] \, d\bar{\xi}$$

The compatibility relations of two-dimensional steady supersonic flow can be expanded likewise. Realizing that instead of ξ and η one can introduce

$$x_0 = \frac{1}{2} (\xi + \eta)$$

$$z_0 = \frac{1}{2} (\eta - \xi)$$

one obtains

$$(1 - M_0^2) \frac{\partial u_1}{\partial x_0} + \frac{\partial v_1}{\partial z_0} = 0$$

$$\frac{\partial u_1}{\partial z_0} - \frac{\partial v_1}{\partial x_0} = 0$$

These equations are of the same form as those of the conventional linearized theory. The important difference, however, is that they contain as independent variables x_0 and z_0 rather than the physical coordinates x and z. The new theory therefore is an analytical characteristics theory. Oswatitsch has also shown that in the three-dimensional and in the unsteady case the equations are again the same as those of the conventional linearized theory except x, y, z and the time t must be replaced by x_0, y_0, z_0 and t_0. In this more general case, however, the integrations must be carried out along the so-called "bi-characteristics" in order to find the position of the disturbed Mach surfaces.

Oswatitsch and collaborators applied this theory to several problems that presented considerable difficulties up to now. Figure 5 shows computations of steady supersonic flow past a delta wing with near-sonic leading edges [4]. The new method makes it possible to determine the shock position (dotted line). Figure 6 shows the flow fields and shock waves which are generated by a body of revolution as it decelerates from supersonic through sonic to subsonic flight speeds [5]. Figure 7 finally demonstrates the application of this method to steady supersonic flow past a cascade [6]. Again, it is seen that the shock waves can be predicted quite well.

Turning now to the problem of oscillating airfoils in transonic flow, most analyses have been based on the linearized transonic small-disturbance equation which is valid only if the reduced frequency of oscillation is sufficiently high. However, wing and compressor blade flutter often occur in the lower frequency range, thus requiring solutions of the nonlinear transonic small-perturbation equation. Yet, it is always possible to assume the vibration amplitude to be quite small. Therefore, the total perturbation potential can be written as the sum of the potentials

$$\Phi(x,y,z,t) = \Phi_1(x,y,z) + \Phi_2(x,y,z,t)$$

so that Φ_1 accounts for the steady flow and Φ_2 for the unsteady flow component. This allows the nonlinear perturbation equation to be split into a nonlinear steady-state equation

$$(1 - M_\infty^2)\,\Phi_{1xx} - M_\infty^2\,(\gamma+1)\,\Phi_{1x}\,\Phi_{1xx} + \Phi_{1yy} + \Phi_{1zz} = 0$$

and into an unsteady equation

$$(1 - M_\infty^2)\,\Phi_{2xx} - M_\infty 2\,(\gamma + 1)\,\frac{\partial}{\partial x}\,(\Phi_{1x} + \Phi_{2x}) + \Phi_{2yy} + \Phi_{2zz} - 2M_\infty^2\,\Phi_{2xt} - M_\infty^2\,\Phi_{2tt} = 0$$

which is linear in Φ_2 but contains variable coefficients because of the presence of Φ_1. An approximate solution of this equation was proposed by Teipel [7] and Hosokawa [8] for oscillating airfoils in two-dimensional flow at $M_\infty = 1$. Assuming harmonic time dependence we have for

$$\Phi_2(x,z,t) = \psi(x,z)\, e^{ikt}$$

and hence the two equations

$$-(\gamma + 1)\,\Phi_{1x}\,\Phi_{1xx} + \Phi_{1zz} = 0$$

$$-(\gamma + 1)\,\Phi_{1x}\,\psi_{xx} - (\gamma + 1)\,\Phi_{1xx}\,\psi_x + \psi_{zz} - 2ik\,\psi_x + k^2\,\psi = 0$$

In 1955, Oswatitsch proposed to assume Φ_{1xx} to be a constant which reduces the steady transonic flow problem to the solution of a parabolic differential equation, namely

$$\Phi_{1xx} - \Gamma\,\Phi_{1x} = 0$$

where

$$\Gamma = (\gamma + 1)\, \mathring{\Phi}_{1xx} = \text{Const} > 0$$

Generalizing Oswatitsch's idea to the oscillatory case, Teipel put

$$-(\gamma + 1)\, \Phi_{1x}\, \psi_{xx} - \Gamma\psi_x + \psi_{zz} - 2ik\,\psi_x + k^2\, \psi = 0$$

Neglecting the first term he arrived at a new parabolic differential equation

$$\psi_{zz} - A\psi_x + k^2\, \psi = 0$$

where

$$A = \Gamma + 2ik$$

whose solution can easily be found. A comparison of this theory with experiments is shown in Figure 8 and is seen to be quite good. Hosokawa [8] further refined this theory by adding a correction function which allows the prediction of transonic shocks. We have recently applied this Teipel-Hosokawa approach to transonic flow past oscillating bodies of revolution and found generally good agreement with available experiments [9].

A second approach to the unsteady transonic flow problem has been proposed by Coupry & Piazzoli and by Eckhaus [10]. Since the interaction of small unsteady perturbations with transonic shock waves may be an important mechanism to induce flutter,the aforementioned authors proposed to simplify the typical transonic flow field shown in Figure 9a by the model of Figure 9b. The resulting boundary-value problem thus becomes mathematically tractable;and calculations by Eckhaus,which are valid only at higher frequencies,indicate indeed an important destabilizing mechanism.

We have recently investigated another wave interaction problem, namely oscillatory supersonic tunnel interference [11]. Figure 10 shows two-dimensional supersonic flow past an oscillating airfoil or panel which is mounted in a wind tunnel. Assuming small amplitudes of oscillation the linearized equations were used, i.e.

$$\frac{2}{\gamma-1}\frac{\partial c}{\partial t} + \frac{2}{\gamma-1}\, u_\infty \frac{\partial c}{\partial x} + c_\infty \frac{\partial u}{\partial x} + c_\infty \frac{\partial w}{\partial z} = 0$$

$$\frac{\partial u}{\partial t} + u_\infty \frac{\partial u}{\partial x} + \frac{2}{\gamma-1}\, c_\infty \frac{\partial c}{\partial x} = 0$$

$$\frac{\partial w}{\partial t} + u_\infty \frac{\partial w}{\partial x} + \frac{2}{\gamma-1}\, c_\infty \frac{\partial c}{\partial z} = 0$$

where u and w are the flow components and c the velocity of sound. This system of equations was solved by a numerical characteristics method and pressure distributions were computed for solid, porous and slotted tunnel interference. The generalization of this method to oscillating cascades is now under investigation.

Finally I would like to say a few words about a new oscillatory pressure measuring technique which was first developed at the National Aerospace Laboratory in Amsterdam [12] and with which we have recently worked ourselves [13]. This technique allows the measurement of detailed oscillatory pressure distributions at very reasonable cost since only one transducer is required. The basic setup is shown in Figure 11. Pressure tubes connect the model's pressure holes via a scannivalve with the transducer which is located outside the tunnel. After determination of the transfer function of the tube-scannivalve configuration the pressures recorded by the transducer can easily be related to the actual pressures on the model surface. Bergh [14] obtained detailed pressure distributions on airfoil-oscillating control surface combinations in transonic flow with this technique. Its application to flow problems in turbomachines should yield valuable information.

References

1. Spee, B. M., "Wave propagation in transonic flow past two-dimensional airfoils", NLR-TN T. 123,1966, National Aerospace Laboratory, Amsterdam.

2. Lin, C. C., E. Reissner, H. S., Tsien; "On Two-Dimensional Non-steady Motion of a Slender Body in a Compressible Fluid", J. Math. and Phys., Vol. XXVII, No. 3, pp. 220-231.

3. Oswatitsch, V., "Ausbreitungsprobleme", ZAMM Vol. 45, No. 7/8, 1965.

4. Sun, E. Y. C., "Nicht Angestellte Deltaflugel mit Unterschall-und Schallvorderkanten", Journal de Mecanique, No. 3, 1964.

5. Stuff, R.; "Analytische Berechnung Von Verdichtungsstoessen beschleumigter oder verzogerter Rotationskoerper", DLR - Forschungsbericht No. 68-62, 1968.

6. Starken H. and H. Lichfuss, "Supersonic Cascade Performance", Lecture Series No. 39 on Advanced Compressors, AGARD - LS - 39 - 70, 1970.

7. Teipel, I.; "Die instationaren Luftkrafte bei der Machzahl Eins", S. F. Flugwissenschaften 12 (1964), pp. 6-14.

8. Hosokawa, I.; "A simplified analysis for transonic flows around thin bodies", Symposium Transsonicum, Aachen 1962.

9. Liu, D. D., M. F. Platzer, S. Y. Ruo; "On the Calculation of Static and Dynamic Stability Derivatives for Bodies of Revolution at Subsonic and Transonic Speeds", AIAA Paper No. 70-190, 8th Aerospace Sciences Meeting, New York, 1970.

10. Eckhaus, W.; "Two-dimensional Transonic Unsteady Flow with Shock Waves", AFOSR TR No. 59-591, 1959.

11. Platzer, M. and C. V. Pierce; "Theoretical Investigation of Wind Tunnel Interference Effects Upon Fluttering Panels and Airfoils in Low Supersonic Flow", Proc. of AIAA/ASME Structures, Structural Dynamics and Materials Conference, Denver, Colorado, April 22-24, 1970.

12. Bergh, H. and H. Tijdeman; "Theoretical and Experimental Results for the Dynamic Response of Pressure Measuring Systems", NLR-TR F.238, January 1965, National Aerospace Laboratory, Amsterdam.

13. Simmons, J. M. and M. F. Platzer; "Experimental Investigation of Incompressible Blow past Airofils with Oscillating Jet Flaps", AIAA paper No. 70-79, 8th Aerospace Sciences Meeting, New York, 1970

14. Tijdeman H. and H. Bergh; "Analysis of Pressure Distributions on a Wing with Oscillating Control Surface in Two-dimensional High Subsonic and Transonic Flow", NLR-TR F.253, March 1967, National Aerospace Laboratory, Amsterdam

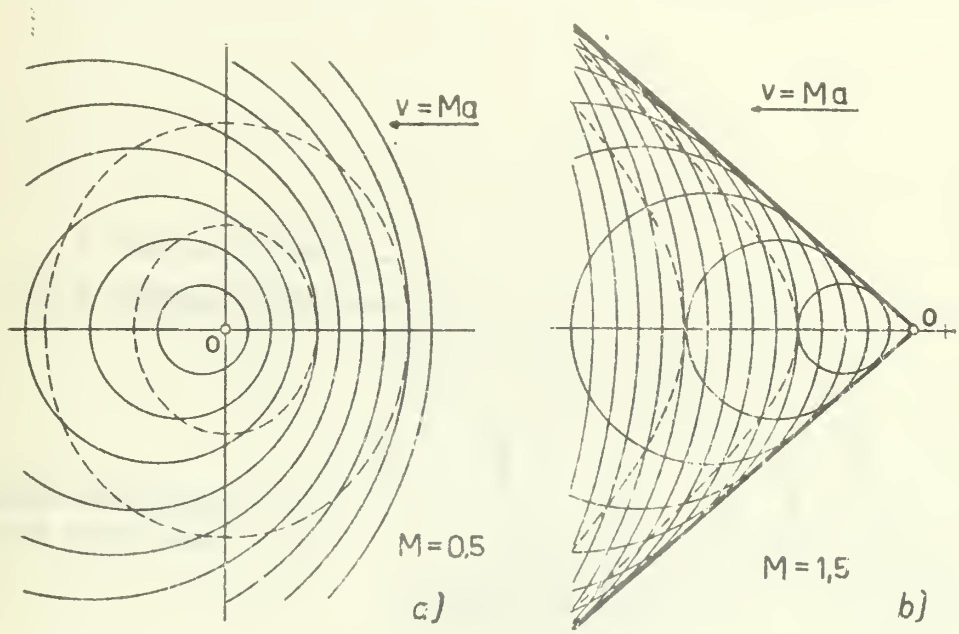


Fig. 1 Source in Subsonic and Supersonic Flow

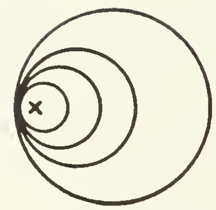


FIGURE 2a ACOUSTIC SOURCE IN TRANSONIC FLOW.

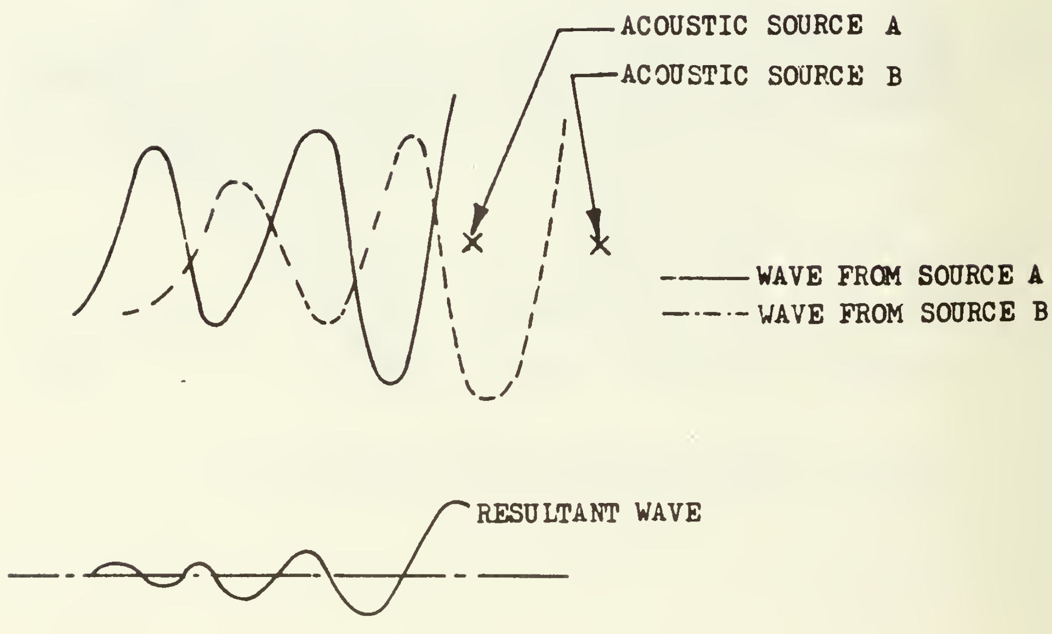


FIGURE 2b CANCELLATION OF RECEDING WAVES.

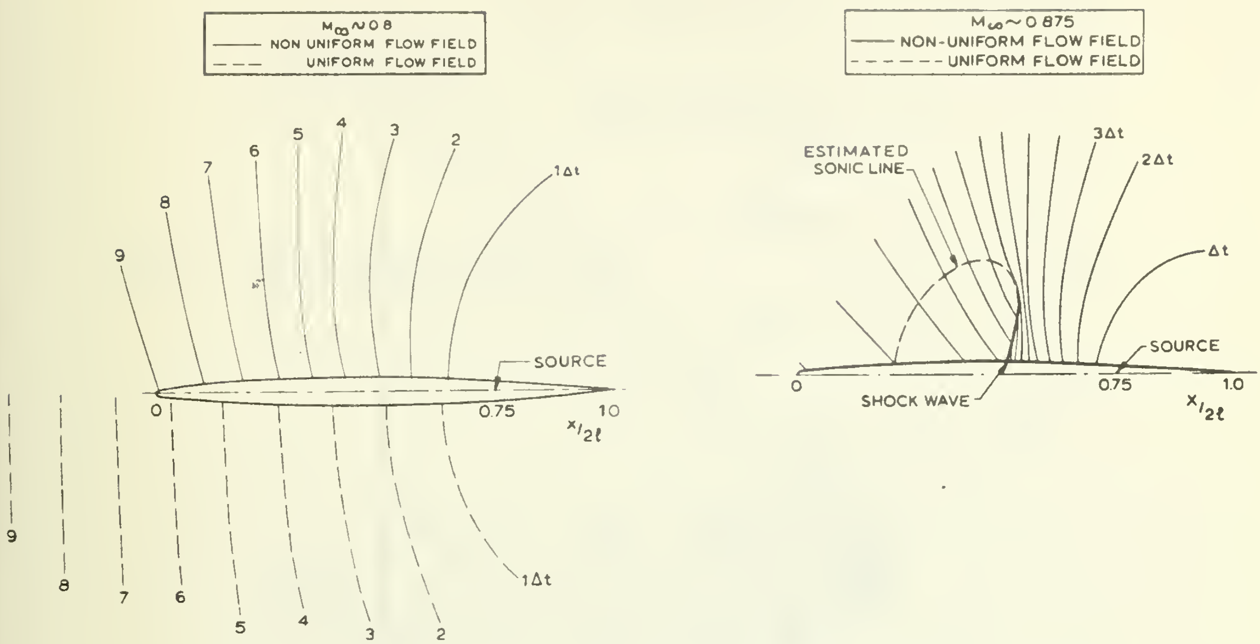


Fig. 3 Wave Propagation in Non-Uniform and Uniform Basic Flow (Ref. 14)

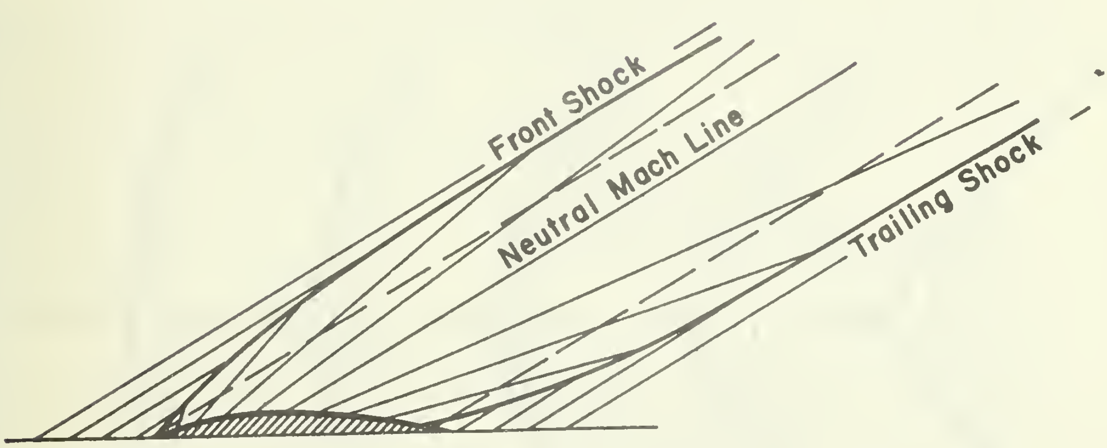


FIGURE 4. TWO-DIMENSIONAL STEADY SUPERSONIC FLOW AROUND SLENDER PROFILE

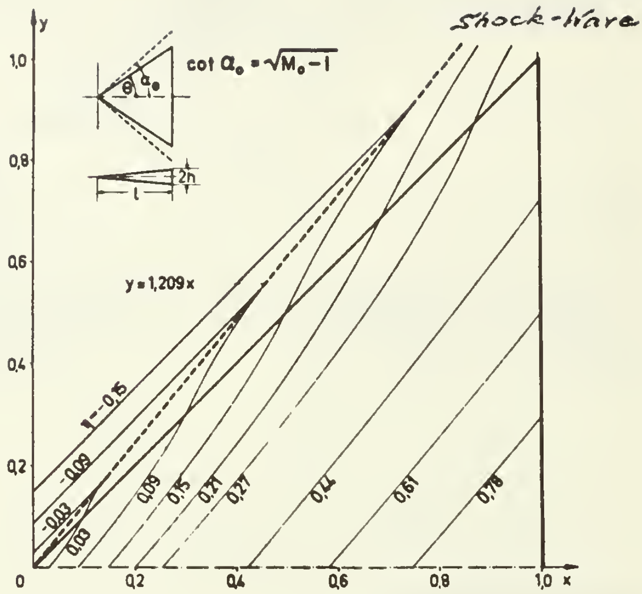


Fig. 5 Rhombic Delta Wing with 16 Percent Thickness Ratio and Sonic Leading Edge (Ref. 4)

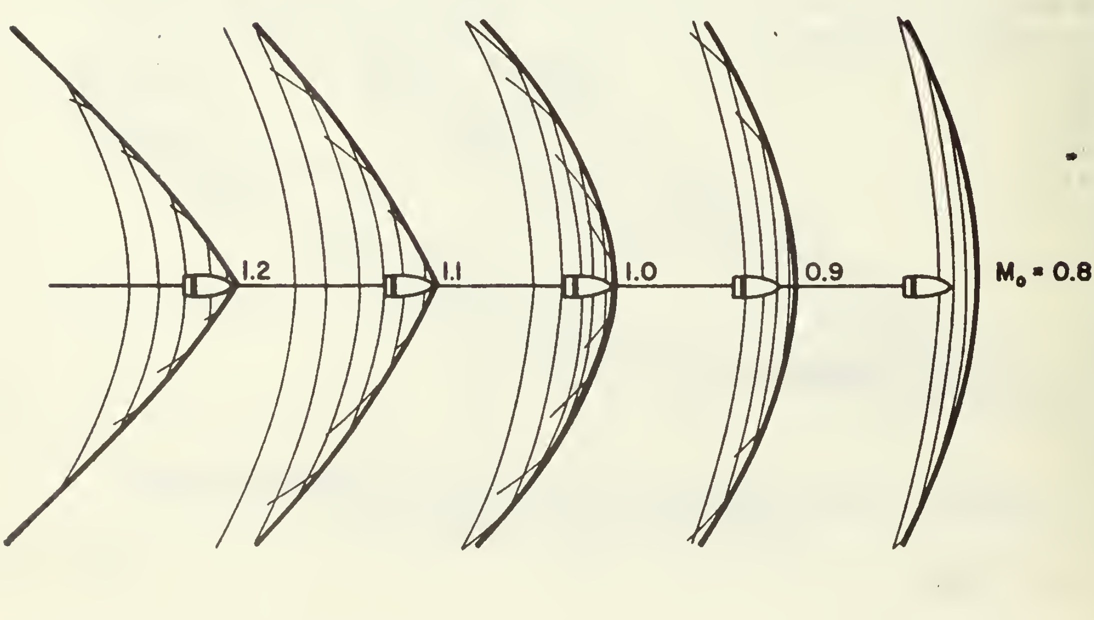


Fig. 6 Decelerating Body with Separating Shock Wave near Sonic Speed (Ref. 5)

Fig. 7 Steady Supersonic Flow Past Cascade (Ref. 6)

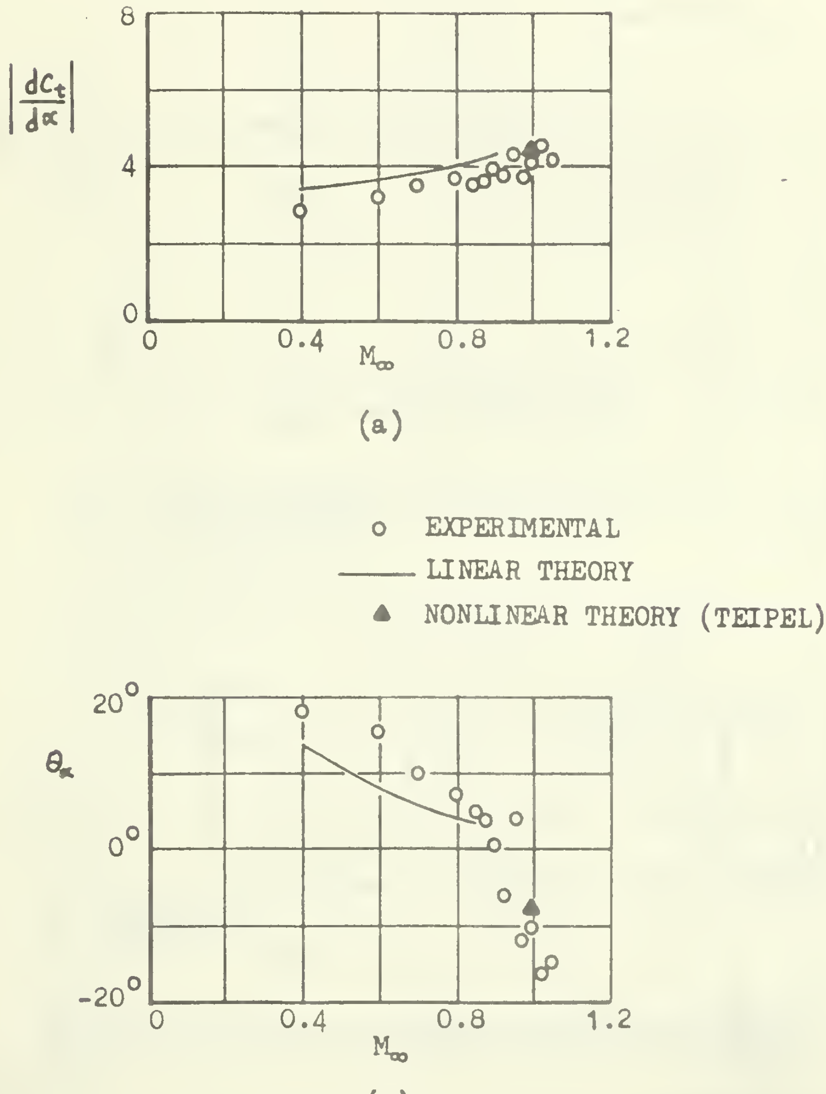
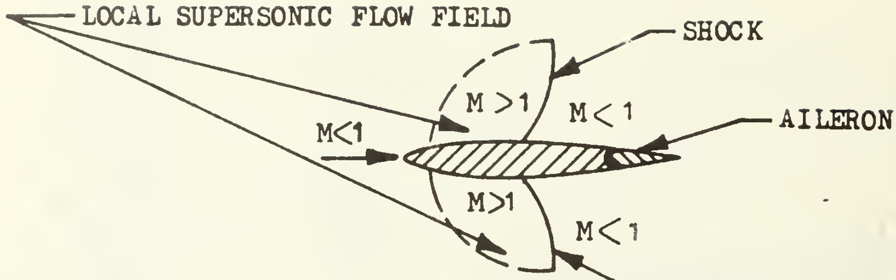
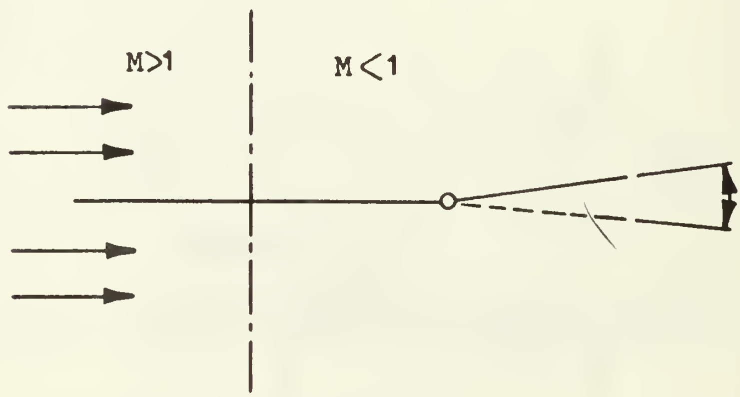


Fig. 8 Magnitude and Phase of Lift Curve Slope for Oscillating Airfoil in Transonic Flow (Refs. 7 and 8)



(a) TYPICAL TRANSONIC FLOW FIELD



(b) ECKHAUS' FLOW MODEL

Fig. 9 Simplified Transonic Flow Model (Ref. 10)

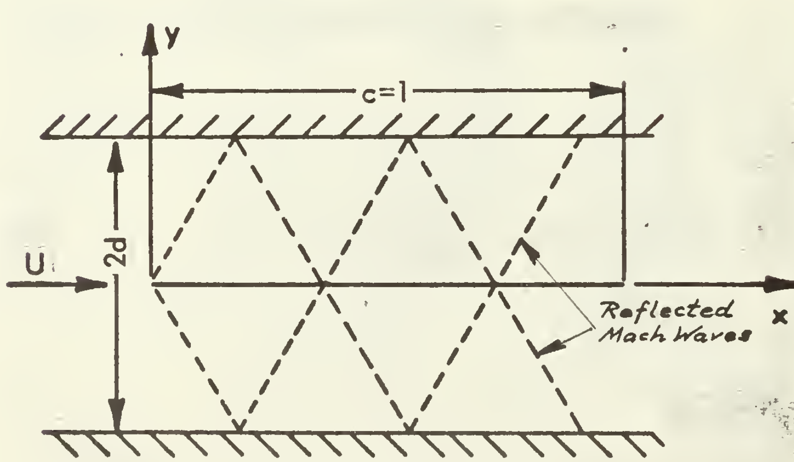


Fig. 10 Oscillating Airfoil in Two-Dimensional Wind Tunnel

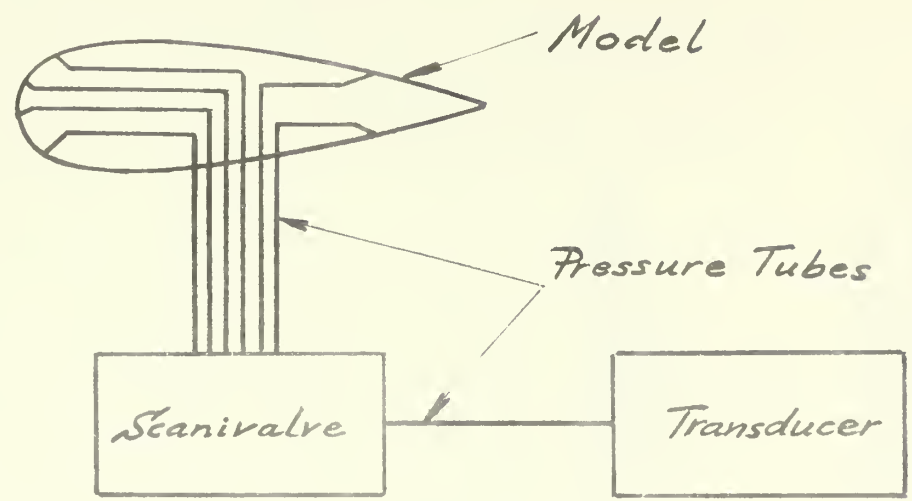


Fig. 11 Oscillatory Pressure Measuring Technique (Refs. 12 and 14)

SOME RECENT DEVELOPMENTS IN THE
NUMERICAL ANALYSIS AND SIMULATION
OF FLUID TURBULENCE

by

Dr. T. H. Gawain
Professor
Naval Postgraduate School

CONTENTS

FIGURES

1.0 Introduction

The great majority of flows in turbomachines are highly turbulent in character. Indeed, most flows of technical and scientific importance are turbulent rather than laminar. Hence the phenomena of turbulence have been the subject of persistent research, both experimental and theoretical, for well over a century. Yet despite the enormous effort devoted to their study, these phenomena are still very inadequately understood. In fact, fluid turbulence remains one of the major unsolved problems of theoretical mechanics.

The reason for this lack of progress is the tremendous complexity which characterizes turbulence. In the first place, the basic equations of motion are nonlinear. If this were not the case, the motion could be split into independent Fourier components which could then be studied separately. However, because of the non-linearity, every component interacts with and is influenced by every other component. Moreover, the fluctuations in a turbulent flow field have a four-dimensional character. They encompass three spatial dimensions and one time dimension. Within each one of these four dimensions, the fluctuations span very broad bands of wave number and frequency. Consequently, the overall process involves myriads of degrees of freedom, all intricately and inextricably coupled together.

Fortunately, the tremendous development of the modern digital computer in recent years has now provided a powerful new tool for simulating and investigating the basic phenomena of turbulence by numerical methods. At the Naval Postgraduate School we have begun exploring this relatively new approach to the turbulence problem and, as you will see presently, we have some very interesting and encouraging results to report.

Of course, at this early stage our research efforts are devoted primarily toward establishing a better understanding of the fundamental phenomena of turbulence. Therefore we are not immediately concerned with the specific applications to turbomachinery or to other particular technical uses. However, since turbulence does play a crucial role in determining the performance of all types of turbomachines, any progress which is made toward better understanding of turbulence fundamentals will sooner or later be reflected in the design of turbomachines and other technical devices. Hence a review of some recent developments in this relatively new branch of numerical fluid dynamics research should be of interest and significance to specialists in turbomachinery, particularly in a conference such as the present one, which is oriented toward assessing the potentialities for the future development of turbomachines.

At the Naval Postgraduate School, we have begun exploring the basic turbulence problem, with the aid of the computer, from two complementary points of view.

On the one hand, we have developed a method for solving the unsteady equations which govern the detailed fluctuating motion. In principle, these equations, along with the appropriate boundary conditions, suffice to determine the turbulent motion in complete detail. However, the number of degrees of freedom which would be needed to perform such a calculation on a three-dimensional space grid would be overwhelmingly large. Consequently, if we wish to work with the detailed unsteady equations, it is necessary at the present stage of computer technology to limit the analysis to a purely two-dimensional basis. Unfortunately, this limitation admittedly does impair the physical realism of the numerical model. For example, the physically important mechanism of vortex stretching cannot be simulated in the ordinary two-dimensional treatment. Also, the energy spectrum is definitely affected. Hence, the two-dimensional methods cannot be expected to give close quantitative agreement with experimental results. However, despite these limitations, this model does simulate certain features of true turbulence in at least a qualitative fashion. For example, it exhibits the non-linearity and apparent randomness of the real phenomena. Also, the two-dimensional analysis is known to portray quite well the early stages of instability and transition from laminar to turbulent flow. Hence, the two-dimensional model is a useful, if imperfect, research tool. Despite its limitations, this model should help shed some light on the phenomena we are seeking to understand.

On the other hand, we have also tackled the turbulence problem by the more conventional method of working not with the detailed equations of the fluctuating motion, but only with the ensemble averaged equations. This approach has the merit that it vastly reduces the computational burden, so that we may retain a fully three-dimensional treatment. However, because of the nonlinearity of the equations, the averaging process leads to a genuine loss of essential information. Consequently, the ensemble averaged equations do not in themselves constitute a determinate set. In particular, these equations do not provide the information needed to fix the turbulent Reynolds stresses. This basic indeterminacy of the ensemble averaged equations is usually referred to as the closure problem of turbulence theory.

In order to provide a closed and determinate set of equations, it is therefore clearly necessary to go beyond the ensemble averaged equations alone. Whenever actual solution of the complete equations for the detailed fluctuating motion is ruled out as impractical, it becomes unavoidably necessary to substitute instead some auxiliary hypotheses of an empirical and heuristic nature. In particular, the auxiliary hypotheses must suffice to fix the Reynolds stresses which would otherwise remain indeterminate. Naturally, since these hypotheses are used in lieu of a much more detailed and involved solution for the fluctuating motion, they constitute a simplification and a phenomenological approximation of the actual phenomena. The question that arises is whether suitable heuristic hypotheses can be found which are simple enough to be practically calculable, general enough to fit a wide range of conditions, and accurate enough to provide acceptable agreement with experiment. Unfortunately, there is no conclusive way to tell in advance just what set of hypotheses, if any, will meet these criteria. However, a set of such hypotheses have been formulated which do appear rather promising. These hypotheses, and some typical results computed with their aid, are reviewed in this paper. While our research is far from complete, the results obtained so far are very encouraging.

In the remainder of this paper we first review some of the key concepts and typical results relating to the heuristic model of turbulence based on the three-dimensional ensemble averaged equations. Then we consider some typical results obtained by solving the equations for the detailed fluctuating motion, although only on a two dimensional basis.

We believe that these two representations, each of which has its own characteristics limitations, serve to complement each other. Our hope and expectation is that these two avenues of research, now distinct, will ultimately converge, providing us with a deeper insight into the basic phenomena of fluid turbulence.

2.0 Basic Principles of the Unified Heuristic Model

The key equations which comprise the heuristic turbulence model are summarized in this section, and significant features of these equations are briefly noted.

2.1 Ensemble Averaged Equation of Continuity

$$\left(\frac{\partial U_i}{\partial x_i}\right) = 0 \qquad i = 1, 2, 3 \qquad (2.1\text{-}1)$$

Of course U_i is the mean velocity component along coordinate axis x_i.

2.2 Ensemble Averaged Equations of Motion (The Navier Stokes Equations)

In the process of averaging the equations of motion over the ensemble, the terms which are quadratic in the perturbation velocities give rise to the so-called Reynolds stress tensor,

$-\rho\,\overline{u_i u_j}$, where the overbar designates the ensemble average. Since we are here concerned only with incompressible flows, it is convenient to divide the equations of motion through by density. This reduces the Reynolds stresses to the purely kinematic form $-\overline{u_i u_j}$. The first invariant of this tensor has the character of a kinematic pressure (pressure divided by density) and may be designated as the turbulence pressure P_t. Moreover, this quantitly is proportional to the kinetic energy E of the turbulence. Thus, by definition,

$$P_t = \frac{1}{3}\,\overline{u_i u_i} = \frac{1}{3}\,(\overline{u_1^2} + \overline{u_2^2} + \overline{u_3^2})$$

$$\text{and} \quad E = \frac{1}{2}\,\overline{u_i u_i} = \frac{1}{2}\,(\overline{u_1^2} + \overline{u_2^2} + \overline{u_3^2}) \tag{2.2-1}$$

whereupon

$$P_t = \frac{2}{3}\,E \tag{2.2-2}$$

The turbulence kinematic pressure P_t can be eliminated as a separate unknown of the problem by lumping it in with the ordinary pressure p (divided by density ρ). Thus we define a resultant kinematic pressure P by the expression

$$P = \left(\frac{p}{\rho} + \frac{2}{3}\,E\right) \tag{2.2-3}$$

As a result of the above step, the only Reynolds stresses which remain in the problem are the deviatoric components τ_{ij} as defined by the expression

$$\tau_{ij} = \left(-\overline{u_i u_j} + \delta_{ij}\,\frac{2}{3}\,E\right) \tag{2.2-4}$$

Although there are six distinct components of τ_{ij}, it follows from the definition (2.2-4) that

$$\tau_{ij} = \tau_{11} + \tau_{22} + \tau_{33} = 0 \qquad (2.2\text{-}5)$$

Consequently, these six residual Reynolds stresses now embody just five independent degrees of freedom.

With this notation the equations of motion become

$$\left(\frac{\partial U_i}{\partial t}\right) + \frac{\partial}{\partial x_j}\left(U_i U_j\right) = -\left(\frac{\partial P}{\partial x_i}\right) + \nu\left(\frac{\partial^2 U_i}{\partial x_j \partial x_j}\right) + \left(\frac{\partial \tau_{ij}}{\partial x_j}\right) \qquad (2.2\text{-}6)$$

Of course ν is the ordinary molecular kinematic viscosity.

In the case of laminar flow, the Reynolds stresses τ_{ij} vanish; and Eqs. (2.1-1) and (2.2-6), along with the appropriate boundary conditions, then suffice to determine the detailed solution for any particular case.

For turbulent flow, however, these equations do not suffice, because of the presence of the as yet indeterminate Reynolds stresses τ_{ij}. Additional relations must therefore be sought sufficient to fix these stresses. However, the additional relations which are usually involved for this purpose generally introduce further unknowns, so that the formulation ultimately becomes quite complex.

2.3 Mixing Length

The mean flow strain rate tensor at a point is defined by the expression

$$\Gamma_{ij} = \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) \qquad (2.3\text{-}1)$$

A corresponding generalized shearing strain rate may be defined by the relation

$$\Omega^2 = \frac{1}{2} \; \Gamma_{ij} \; \Gamma_{ij} \tag{2.3-2}$$

The quantities Ω^2 and Ω are true scalars. Their values are independent of the orientation of the axes with respect to which the tensor components Γ_{ij} happen to be expressed.

It can be shown that for the special case of parallel flow in a uniform channel, the quantity Ω reduces simply to the derivative $\left(\frac{dU}{dy}\right)$ of the velocity profile U with respect to distance y from the wall. Hence Ω may be interpreted as a generalization of this simple velocity derivative.

Since Ω^2 is continuous, we can define the following derived quantity which proves to be significant for subsequent calculations, namely,

$$|\Omega \nabla \Omega|^2 = \frac{1}{4} \left(\frac{\partial \Omega^2}{\partial x_i} \right) \left(\frac{\partial \Omega^2}{\partial x_i} \right) \tag{2.3-3}$$

The flow field as a whole has an overall length scale λ_o which can be defined by the relation

$$\lambda_o^{\,2} = \frac{\int_{\text{all space}} \Omega^4 dv}{\int_{\text{all space}} |\Omega \nabla \Omega|^2 dv} \tag{2.3-4}$$

The separate volume integrals in this expression may be finite or infinite, but their quotient remains finite in the limit.

Associated with each point $\bar{x}$ of the flow field is a local mixing length $\lambda(\bar{x})$. However, the value of λ at point $\bar{x}$ is dependent to some extent on conditions at all neighboring points $\bar{x}'$. The influence of conditions at variable point $\bar{x}'$ on conditions at the reference point $\bar{x}$ are assumed to depend on the spatial separation

$$\bar{\xi} = \bar{x}' - \bar{x} \tag{2.3-5}$$

according to a Gaussian weighting function of the form

$$w(\xi) = C\, e^{-\frac{\xi^2}{\lambda_o^2}} \tag{2.3-6}$$

The constant C which fixes the overall amplitude of this weighting function is chosen so as to satisfy the simple normalizing condition

$$\int_{\text{all space}} w(\xi)dv^1 = C \int_{\text{all space}} e^{-\frac{\xi^2}{\lambda_o^2}}\, dv' = 1 \tag{2.3-7}$$

In relation to any specified reference point $\bar{x}$, we can now define weighted averages of the quantities Ω^4 and $|\Omega\nabla\Omega|^2$ according to the following rules. It is convenient however to assign new symbols I^2 and J^2 to these weighted averages as follows.

$$I^2(\bar{x}) = \int_{\text{all space}} w(\xi)\, \Omega^4(\bar{x}')\, dv' \tag{2.3-8}$$

$$J^2(\bar{x}) = \int_{\text{all space}} w(\xi)\, |\Omega\nabla\Omega|^2(\bar{x}')\, dv' \tag{2.3-9}$$

Finally the required mixing length λ at point $\bar{x}$ is defined simply by the ratio

$$\lambda^2(\bar{x}) = \frac{I^2(\bar{x})}{J^2(\bar{x})} \tag{2.3-10}$$

While it is perhaps not immediately evident from the foregoing relations expressed in their present form, it nevertheless turns out that the mixing length λ as defined above is closely related to the well-known mixing length of von Karman. However, von Karman's mixing length is defined solely in terms of the conditions at the point $\bar{x}$ itself, whereas the present mixing length λ is defined on the basis of weighted average conditions in the general vicinity of the point. Also von Karman's definition applies only to parallel flow near a wall while the present definition applies also to regions far from any fixed boundary.

2.4 Eddy Viscosity

The eddy viscosity ϵ at any point of the flow field is defined by the expression

$$\epsilon = \frac{\frac{1}{2}\,\tau_{ij}\,\Gamma_{ij}}{\frac{1}{2}\,\Gamma_{ij}\,\Gamma_{ij}} \tag{2.4-1}$$

The denominator of this fraction will be recognized as the scalar quantity Ω^2. The numerator is also a true scalar which represents the rate at which the mean flow does work against the resistance of the Reynolds stresses. Actually this measures the rate of generation of turbulent energy at the point. Note that eddy viscosity ϵ has the general character and dimensions of stress divided by strain rate and that it too is a true scalar.

It should be mentioned that some theorists question whether Reynolds stresses can be related to strain rates in any useful way through the use of an eddy viscosity concept, or even whether an eddy viscosity can be truly said to exist as a real property of a turbulent flow field. The answer to this question depends of course on how the eddy viscosity is defined and used. However, there can be no doubt that, as defined above, the eddy viscosity ϵ exists as a true scalar and is in every sense just as real and definite a property of the flow field as are the Reynolds stresses and the mean-flow strain rates themselves. The fact that the numerical magnitude of the eddy viscosity, like that of Reynolds stresses themselves, often happens to be unknown does not in any way invalidate this conclusion.

2.5 Stress/Strain Rate Law

In incompressible flow, the deviatoric components of the mean-flow strain rate tensor are simply the tensor components Γ_{ij} as previously defined by Eq. (2.3-1). These strain rates give rise to corresponding deviatoric viscous kinematic stresses $(\tau_{ij})_v$ (stress divided by density) according to the simple proportionality

$$(\tau_{ij})_v = \nu \, \Gamma_{ij} \tag{2.5-1}$$

where ν is the ordinary molecular kinematic viscosity.

Naturally, it is tempting to assume that the deviatoric Reynolds stresses can be expressed in the corresponding form

$$\tau_{ij} = \epsilon \, \Gamma_{ij} \tag{2.5-2}$$

where ϵ is the eddy viscosity as defined in Eq. (2.4-1).

From the theoretical viewpoint, however, there is no particular reason why Eq. (2.5-2) should necessarily be exactly true. We must be prepared to accept a more general relationship, if necessary. The most complex state that could exist at any point can always be expressed, with complete generality, in the form

$$\tau_{ij} = \epsilon \left[\Gamma_{ij} + \Omega f_{ij}\right] \tag{2.5-3}$$

The dimensionless tensor quantities f_{ij} in this relation express the non-isotropic aspects of the stress/strain rate relation at the point under consideration. They can be shown to satisfy the constraints

$$\begin{aligned} f_{ij} &= f_{ji} \\ f_{ii} &= 0 \\ f_{ij}\, \Gamma_{ij} &= 0 \end{aligned} \tag{2.5-4}$$

Eq. (2.5-3) shows that in order to be able to predict the Reynolds stresses τ_{ij} and to correlate them with the mean flow strain rates Γ_{ij}, it is theoretically necessary to establish independently the quantities ϵ and f_{ij}. However, in comparison with ϵ, the f_{ij} are relatively unimportant, for it can be shown that the Reynolds stress increments associated with the f_{ij} make no net contribution to the generation of turbulent energy. Moreover, while the experimental evidence on this point is meager, such data as are available suggest that the f_{ij} are in fact negligible everywhere except possibly in the immediate vicinity of a wall. For these various reasons we assume for the present model simply that

$f_y = 0$, whereupon Eq. (2.5-2) becomes an acceptable, if approximate, statement of the relevant stress/strain rate law.

2.6 Heuristic Eddy Viscosity Coefficient

Even with the simplification that $f_{ij} = 0$, Eq. (2.5-3) cannot be used to fix the Reynolds stresses τ_{ij} unless and until the eddy viscosity ϵ be independently expressed in terms of known or computable features of the flow field.

For the purpose of establishing ϵ, the following empirical function has been found to give reasonable overall agreement with experimental results, as will be illustrated later.

$$\frac{\epsilon}{\lambda\sqrt{2E}} = \alpha = 0.065\left\{1 + e^{-\left(\frac{y}{\lambda} - 1\right)^2}\right\} \qquad (2.6\text{-}1)$$

In this relation, y represents distance to the nearest fixed wall, if any. Far from any wall, y becomes infinite, and the dimensionless eddy viscosity coefficient α reduces to a simple constant. On the other hand, as we approach close to a wall, y goes to zero but the ratio $\frac{y}{\lambda}$ approaches unity in the limit. Hence at the wall itself the exponent $\left(\frac{y}{\lambda} - 1\right)^2$ simply vanishes.

2.7 Ensemble Averaged Energy Equation

The foregoing relations (2.5-3) and (2.6-1) still cannot be solved for eddy viscosity ϵ and Reynolds stress τ_{ij} until the distribution of the turbulent energy E can be independently expressed. An independent energy equation can be obtained for this purpose by integrating and ensemble averaging the equations of motion. The result can finally be reduced to the following form.

$$\left(\frac{\partial E}{\partial t}\right) = \epsilon\,\Omega^2 - \frac{\partial}{\partial x_i}\left(U_i E\right) - \frac{\partial}{\partial x_i}\left[\gamma\,\epsilon\left(\frac{\partial E}{\partial x_i}\right)\right] - \dot{Q} \qquad (2.7\text{-}1)$$

The terms on the right represent, respectively, generation of turbulent energy, convection, turbulent diffusion and dissipation of kinetic energy into heat. Unfortunately, this equation introduces two additional unknowns, namely the dissipation $\dot{Q}$ and the dimensionless factor γ in the diffusion term. The factor γ represents the ratio of the diffusion coefficient to the kinematic eddy viscosity ϵ.

2.8 Heuristic Dissipation and Diffusion Coefficients

The following empirical functions have been found to provide reasonable estimates of dissipation and diffusion, respectively.

$$\frac{\dot{Q}}{(2E)^{7/6}\,J^{1/3}} = \beta = \frac{1}{3.7}\left\{1 + e^{-\left(\frac{y}{\lambda} - 1\right)^2}\right\}^{-1} \qquad (2.8\text{-}1)$$

$$\gamma = \left\{1.4 - 0.4\,e^{-\left(\frac{y}{\lambda} - 1\right)^2}\right\} \qquad (2.8\text{-}2)$$

2.9 Summary and Critique

The foregoing system of equations constitutes a closed set. When combined with the appropriate boundary conditions, they define a determinate solution for any particular case of turbulent flow, including detailed space-time distributions of U_i, P, λ, E, ϵ and τ_{ij} over the entire flow field.

A very important feature of this turbulence model is the fact that a single consistent set of fundamental relations applies to all cases. Only the boundary conditions change from one application to another. This represents a very significant theoretical advance. In the past there was a tendency to develop a separate empirical turbulence model for each separate type of flow configuration, with no very clear connection between the various empirical constants of one configuration and those of another.

The continuity, momentum and energy equations as used here can be rigorously deduced from fundamental principles. The mixing length concept is heuristic, but represents a generalization of the von Karman mixing length, which is a well-established concept. The stress/strain rate law is completely general, but the simplification $f_{ij} = 0$ warrants further experimental investigation. The expressions for the three dimensionless coefficients α, β, and γ are purely empirical and heuristic. The present versions are based on a lengthy trial and error process involving dimensional analysis, speculative assumptions, numerical experiments, and comparison of computed results with available experimental data. The available data were in many respects meager and inadequate. There is certainly room for improvement in the formulation of these three heuristic functions, based primarily on better experimental information.

Nevertheless, as will be seen in the next section, the degree of success achieved is such as to suggest strongly that the present model is basically correct in its main essentials.

3.0 Some Typical Results Computed by the Heuristic Model

In this section we show the results of calculations obtained by application of the heuristic model to two fundamental and widely different cases. These are flow in a uniform channel,Fig. 3-1, and flow in a free turbulent jet, Fig. 3-4. The first example is dominated by wall effects, the second is a completely unbounded flow. Considering the extreme difference in the boundary conditions for these two cases, it is remarkable that our unified heuristic model gives equally good results for both.

For the channel flow, Fig. 3-2 shows turbulent energy, suitably non-dimensionalized, as a function of distance from the wall. Fig. 3-3 shows the corresponding non-dimensional Reynolds stress versus wall distance.

Figs. 3-5 and 3-6 show the corresponding results for the turbulent jet, namely, the non-dimensional energy and Reynolds stress, respectively, plotted against radius.

All of these curves computed from the heuristic model agree with the experimental points as well as the experimental points agree with one another. In the case of the Reynolds stresses the results of the heuristic model as shown by the solid lines agree well in each case with other known theoretical curves as shown by the dashed lines. We conclude that the heuristic model gives results which are as accurate as or more accurate than those that were obtained by experimental measurement.

Further details concerning the computations may be found in Refs. 1 and 2.

4.0 Two-Dimensional Instability and Turbulence for Flow through a Uniform Channel

In this section we consider the solution, on a two-dimensional basis, for the detailed unsteady motion in a uniform channel, which is the same configuration shown earlier in Fig. 3-1. The following notation is employed:

x = streamwise coordinate

y = transverse coordinate

t = time

$F = F(y,t)$ = stream function of mean flow

$U = F_y(y,t)$ = mean velocity

$\Psi = \Psi(x,y,t)$ = perturbation stream function

R = Reynolds number

It is advantageous to eliminate pressure from the formulation by working with the curl of the vector equation of motion, that is, with the vorticity transport equation. This relation can then be resolved into two distinct but coupled equations which pertain, respectively, to the mean flow and to the perturbation flow.

The first of these equations, which governs the gradual evolution of the mean flow under the influence of the averaged perturbation effects, is found to be

$$F_{yyt} = \frac{1}{R} F_{yyyy} + (\overline{\Psi_x \nabla^2 \Psi_y} - \overline{\Psi_y \nabla^2 \Psi_x}) \tag{4-1}$$

The overbars in Eq. (4-1) denote quantities which have been averaged over x. In the present problem, this type of space average over x is the appropriate substitute for the more general concept of an ensemble average.

The second vorticity transport equation governs the detailed unsteady perturbations themselves, and is vastly more complex than the previous relation for the mean flow. It is found to be

$$\nabla^2\Psi_t = \frac{1}{R}\nabla^4\Psi - F_y\nabla^2\Psi_x + F_{yyy}\Psi_x + \left\{(\Psi_x\nabla^2\Psi_y - \Psi_y\nabla^2\Psi_x) - (\overline{\Psi_x\nabla^2\Psi_y} - \overline{\Psi_y\nabla^2\Psi_x})\right\} \tag{4-2}$$

The detailed solution of the two coupled equations (4-1) and (4-2) is further constrained by the following boundary and initial conditions.

Total flow is held constant, therefore,

$$\begin{aligned} F &= +1 \quad \text{at upper wall,} \quad y = +1 \\ F &= -1 \quad \text{at lower wall,} \quad y = -1 \\ \Psi &= 0 \quad \text{at } y = \pm 1 \end{aligned} \tag{4-3}$$

There is no slip at the walls, hence

$$F_y = 0 \quad \text{and} \quad \Psi_y = 0 \quad \text{at } y = \pm 1 \tag{4-4}$$

In principle, initial conditions may be specified arbitrarily. Normally, the initial mean flow is taken to be simple laminar flow corresponding to

$$F = \frac{3}{2}y - \frac{1}{2}y^3 \tag{4-5}$$

This gives the familiar parabolic mean velocity profile

$$U = \frac{3}{2}(1 - y^2) \tag{4-6}$$

The initial perturbation stream function Ψ is appropriately taken to be periodic in x. Consequently, it retains this initially imposed periodicity at all subsequent times. The particular form of the initial perturbation is usually taken to be the unstable eigenfunction of the linearized problem, as explained later. The initial amplitude of the perturbation may be chosen arbitrarily.

Subject to these boundary initial conditions, numerical solution of Eqs. (4-1) and (4-2) then provides a complete and detailed quantitative record of the subsequent evolution of the flow, both in regard to its mean characteristics and in regard to its unsteady fine scale features.

Notice the great contrast between this type of analysis and the more pragmatic method which would seek to work only with mean flow quantities and averaged equations. The pragmatic method would retain Eq. (4-1) but discard Eq. (4-2) as impractical to solve. Instead it would introduce simplified heuristic assumptions intended to provide a plausible basis for the approximate solution of all relevant mean flow and mean turbulence quantities.

4.1 The Linearized Problem

Note that certain terms in Eqs. (4-1) and (4-2) are quadratic functions of the perturbation amplitude. For perturbations of small amplitude, these terms may be neglected, thus linearizing the equations. In this case, Eq. (4-1) may be integrated at once to obtain the familiar parabolic velocity profile for the mean flow as indicated earlier in Eq. (4-6).

In solving the linearized version of Eq. (4-2), it is useful to shift to complex variables. The complex stream function, call it Ψ_c, can be shown to have solutions of the form

$$\Psi_c = \varphi(y)\, e^{i\alpha x + \beta t} \tag{4.1-1}$$

The real constant α is wave number pertaining to spatial oscillations in the streamwise direction. Its value may be arbitrarily assigned. The complex constant β and the complex function φ then remain to be determined.

Upon substituting (4.1-1) into the linearized version of (4-2), one obtains the well known Orr-Sommerfeld equation. It is convenient to introduce the linear operator

$$L^2 = \left(\frac{\partial^2}{\partial y^2} - \alpha^2\right) \tag{4.1-2}$$

whereupon the required relation assumes the form

$$\left\{ \frac{1}{R} L^2 L^2 - (\beta + i\alpha F_y)\, L^2 - i\alpha F_{yyy} \right\} \varphi = 0 \tag{4.1-3}$$

Notice that the substitution (4.1-1) has reduced the basic partial differential equation (4-2) to the ordinary differential equation (4.1-3). This is a tremendous simplification.

The solutions must be such as to satisfy the conditions of zero velocity at the walls, namely,

$$\varphi = 0 \quad \text{and} \quad \varphi_y = 0 \quad \text{at} \quad y = \pm 1 \tag{4.1-4}$$

Solutions of Eq. (4.1-3) can satisfy the boundary conditions (4.1-4) only for certain discrete eigenvalues of the complex constant β. Call these $\beta^{(1)}$, $\beta^{(2)}$, $\beta^{(3)}$, ... $\beta^{(r)}$ .... Associated with each eigenvalue is a corresponding eigenfunction. Call these $\varphi^{(1)}$, $\varphi^{(2)}$, $\varphi^{(3)}$, ... $\varphi^{(r)}$....

Of particular significance is the algebraic sign of the real part of each complex root $\beta^{(r)}$. A positive value represents a perturbation whose amplitude grows exponentially with time, that is, a hydrodynamic instability. For any assigned value of Reynolds number R and axial wave number α, there are an infinite number of discrete eigenvalues $\beta^{(r)}$ and corresponding eigenfunctions $\varphi^{(r)}$. Calculation discloses, however, that at most one of these characteristic solutions is unstable, call it $\varphi^{(1)}$, and then only for certain combinations of R and α.

Systematic calculations enable us to map the region of instability within a contour in the R α plane as shown in Fig. 4.1-1. These contours are found by solving Eq. (4.1-3) numerically, using a one-dimensional finite difference approximation. A striking fact disclosed by this diagram is that the region of apparent instability definitely depends on the fineness of the grid used in the finite difference calculation. At least 200 grid points are required to obtain reasonable accuracy. Reducing the number of grid points reduces the region of apparent instability. If a coarse grid of only 41 points is used, all evidence of the hydrodynamic instability which really exists is entirely lost! This result emphasizes the great importance of using a very fine grid spacing across the channel.

4.2 Extension to the Non-Linear Case

To solve the non-linear problem numerically, a finite difference approximation of Eq. (4-2) was used. Of course, the finite difference grid for this case must now extend not only across the channel but also streamwise. The grid used contains 64 streamwise and 201 transverse stations, or 12,864 mesh points in all! Notice the tremendous increase in complexity that this represents in comparison with the one-dimensional mesh which suffices for the linearized problem.

To check the adequacy of the two-dimensional mesh and of the corresponding finite difference scheme, some preliminary calculations were made with the non-linear terms deliberately omitted from the equation. Under these circumstances, the results computed on the two-dimensional grid should, of course, agree completely with the results obtained from the simpler linearized analysis. This comparison was carried out and the expected agreement was indeed found, thus confirming the validity of the two-dimensional calculations, at least so far as the linear terms are concerned.

The non-linear terms were then restored, and various numerical experiments were carried out. As a rule, the unstable eigenfunction obtained from the linearized analysis was used as the initial perturbation for the two-dimensional non-linear calculation. Various initial amplitudes were assigned to this perturbation. In each case, the resulting evolution of the flow field was systematically calculated on the digital computer. For runs in which the initial amplitude was small, the computed results at first did not differ appreciably from those of the linearized analysis. Gradually, however, as the perturbation grew, the non-linear effects became more and more pronounced.

In connection with these results, it is useful to consider the resolution of the perturbation stream function Ψ, a real variable, into its complex Fourier components φ_n according to the well known Fourier relations

$$\Psi(x,y,t) = \sum_{n=-\infty}^{+\infty} \varphi_n(y,t) e^{+i n \alpha x} \tag{4.2-1}$$

$$\varphi_n(y,t) = \frac{1}{\pi} \int_{-\frac{\pi}{\alpha}}^{+\frac{\pi}{\alpha}} \Psi(x,y,t) e^{-i n \alpha x} dx$$

In the experiments mentioned above, at time $t = 0$, we usually set the first Fourier component $\varphi(y,0)$ in Eqs. (4.2-1) to agree precisely in form with the unstable eigenfunction $\varphi^{(1)}(y)$ of the linearized problem. Of course, the amplitude could be set arbitrarily. All other Fourier components $\varphi_n(y,0)$ were initially set equal to zero. However, because of the non-linear effects involved in this problem, these various Fourier components gradually change not only in amplitude, but also in form.

Since $\varphi_n(y,t)$ is complex, it may be represented in terms of an amplitude $A_n(y,t)$ and a phase angle $\theta_n(y,t)$ according to the relation

$$\varphi_n(y,t) = A_n\, e^{i\,\theta_n} \tag{4.2-2}$$

Fig. 4.2-1 shows the gradual change in amplitude and phase angle of the initial Fourier component for a typical case. The solid lines indicate initial values; the dashed lines indicate values at a later time.

Associated with each wave number n is a corresponding mean turbulent kinetic energy $\overline{E}_n$. This can be computed from the complex stream function components φ_n according to the relations

$$\overline{E}_n(y,t) = \frac{1}{2}\,(\overline{u^2} + \overline{v^2})_n$$

$$= \left\{[\varphi_n]_y\,[\varphi_{-n}]_y + n^2\alpha^2\varphi_n\,\varphi_{-n}\right\} \tag{4.2-3}$$

As indicated by the overbar, the quantity $\overline{E}_n$ defined above is actually a space average over x.

Of course, the energy in the various wave numbers may be summed to find the total energy $\overline{E}$. Thus

$$\overline{E}(y,t) = \sum_{n=1}^{\infty} \overline{E}_n(y,t) \tag{4.2-4}$$

If desired, the above energies can be averaged across the entire channel according to the following definitions

$$\overline{E}_n(t) = \frac{1}{2}\int_{-1}^{+1} \overline{E}_n(y,t)\,dy$$

$$\overline{E}\,(t) = \frac{1}{2}\int_{-1}^{+1} \overline{E}\,(y,t)\,dy \tag{4.2-5}$$

In the experiments described above, all of the turbulent energy is initially in the fundamental mode, n=1. Not only does this total energy grow, but because of the non-linear interactions, it gradually redistributes itself over the other Fourier modes so as to define a complete spectrum.

A typical energy spectrum is shown in Fig. (4.2-2). The data points conform approximately to a relation of the form

$$E_n \approx E_1 \, n^{-3} \tag{4.2-6}$$

Such rapid attenuation of energy with increasing wave number seems to be typical of the two-dimensional turbulence model. True three-dimensional turbulence would be expected to attenuate less rapidly.

Fig. (4.2-3) shows the evolution of the overall energy for three different amplitudes of the initial kinetic energy. All three of these curves appear to be evolving toward a limiting value which is independent of the initial state of the system. This behavior agrees qualitatively with what we know about true physical turbulence. (Incidentally, the energy shown in this diagram is defined as the turbulent energy plus the mean flow energy minus the initial mean flow energy; this particular combination happens to vary more smoothly than the turbulent energy alone.)

Finally, Fig. (4.2-4) shows typical distributions of mean velocity, mean vorticity and turbulent energy across the channel. The energy distribution is qualitatively somewhat similar to those measured experimentally in physical turbulence, but is not concentrated as closely to the wall. The vorticity jump near the wall is qualitatively correct, but is too small. The mean velocity profile differs only slightly from the laminar parabolic profile. This velocity profile is disappointing; it distinctly fails to approach the familiar logarithmic profile so characteristic of physically turbulent flow in a channel.

Further details and results relating to this investigation may be found in Refs. 3 and 4.

4.3 Significance of the Results

Hitherto, hydrodynamic instability has been amenable to analysis solely on the linearized one-dimensional basis. The present results demonstrate that the detailed simulation of hydrodynamic instability in the non-linear range on a two-dimensional basis is now computationally feasible.

On the other hand, the grossly oversimplified assumption of purely plane perturbations appears to be distinctly inadequate for simulating fully developed physical turbulence. This limitation is probably due to the extreme anisotropy of the plane perturbation model.

The logical next step should therefore be to change the mathematical model so as to reduce this extreme anisotropy while still restricting the number of independent spatial coordinates to just two. This seems to be within the bounds of possibility, although consideration of details lies outside the scope of this paper.

However, much useful work can still be accomplished even within the framework of the plane perturbation assumption, despite its obvious limitations. For example, it should be possible to express the non-linear behavior of the system directly in terms of the eigenfunctions of the associated linear equation. This step might permit an adequate description of the instantaneous state of the system in terms of vastly fewer degrees of freedom than the 12,864 that are utilized in the present formulation. Further research along this line could very possibly lead to a tremendous simplification of the calculations and a corresponding increase in their usefulness.

5.0 References

1. "Development of a Unified Heuristic Model of Fluid Turbulence" - T. H. Gawain and J. W. Pritchett. Naval Postgraduate School Report No. NPS-57GN71021A.
2. "A Unified Heuristic Model of Fluid Turbulence" - T. H. Gawain and J. W. Pritchett, Journal of Computational Physics, Vol. 5, No. 3, June 1970, p. 383-405.
3. "A Numerical Investigation of Turbulence in Plane Poiseuille Flow" - G. D. O'Brien, Jr., Ph.D. dissertation, Dept. of Aeronautics, Naval Postgraduate School, Monterey, June 1970.
4. "Numerical Simulation of Transition and Turbulence in Plane Poiseuille Flow" - T. H. Gawain, Proceedings of the Second International Conference on Numerical Methods in Fluid Mechanics, Sept. 1970.

Discussion

Question:

How does your work on the heuristic model differ from that of Bradshaw? He makes a similar assumption.

Answer:

Yes, a number of investigators are working along somewhat similar lines at this time. I regret that I am not prepared just now to make an adequate comparison of these various approaches, but I refer you to an excellent survey that just came out on this very subject by Prof. W. C. Reynolds of Stanford. It is Stanford Report MD-27 entitled "Computation of Turbulent Flows-State-of-the-Art, 1970", dated October 1970.

Question:

One of the most important aspects of your heuristic model, of course, is the way you modelled the diffusion and dissipation terms. I notice that you do this, first of all, with only one length scale. Other people who have done this in the past have found it necessary to use two length scales. One of these is characteristic of the large scale eddies with some memory of flow history, the other length scale is characteristic of the Kolmogorov equilibrium range. I wonder how it is that you manage with one. What are you doing that they are not doing?

Answer:

This is an excellent question, but it is rather difficult to answer briefly because I don't quite know how to summarize in a nutshell the lengthy trial and error process that went into the actual development of our model. We experimented with various alternative hypotheses, including the use of dual scales. Our aim, of course, was not extreme accuracy but rather reasonable accuracy with the minimum possible degree of complication in the model. Therefore we tried to find a plausible rationale for eliminating the second length scale. One way in which this can be done is described in Ref. 2 of our paper. I don't think I can cover this matter here in a way which is both adequate and brief, so I won't attempt it, but I will give you a reprint of the actual reference so that you can go over the written explanation in detail. In any case, the real justification in this context cannot be in the theoretical reasoning, which is at best a mere plausibility argument; it lies rather in the pragmatic test of agreement with experimental data.

Comment from another participant:

I would like to remark that the second length scale is always dependent on the first because the large eddies provide the energy which ultimately reaches the smaller eddies through the cascade process. Therefore, since the smaller length scale is, in principle, a function of the larger scale, it does not seem unreasonable to try to formulate a model in terms of a single length scale.

Answer:

Thank you for that comment, because it answers the question in such clear physical terms. Of course, our second length scale is not explicitly indicated in our model, but is implicit in some of the relations used. The reference I mentioned brings out this relationship.

Question:

I notice that the expression which you use for the length scale involves both the velocity gradient and the second derivative of the velocity. However, in a flow with a uniform velocity gradient - the simplest non-isotropic flow you can have - the second derivative vanishes. What happens to the length scale then?

Answer:

Theoretically, the length scale becomes infinite. Practically, this means that the length scale tends to grow large compared with the size of the region in which the velocity gradient is sensibly uniform.

The physical interpretation of this derives from the fact that the apparently simple case of a uniform velocity gradient is truly simple only if the flow happens to be laminar. For turbulent flow, it is far from simple. In fact, it is not easy to produce experimentally a turbulent shear flow with a sensibly uniform gradient over most of the flow field, and it is probably impossible to produce one which is both uniform and non-vanishing over the entire field. Boundary effects in turbulent flow tend innately to create curvature in the velocity profile, the intensity of which attenuates with increasing distance from the wall. The model uses the intensity of this distortion as an indirect means of sensing wall distance, and hence also length scale. Thus in a uniform gradient, the length scale is infinite. This amounts to saying that the boundaries which actually fix the length scale must be indefinitely far away. Qualitatively, this is correct.

A more severe limitation in the logical self-consistency of the model, in its present form, is that it does not rely exclusively on such indirect sensing of wall distance. In the heuristic functions governing eddy viscosity, dissipation and diffusion, wall distance now appears explicitly. It would be desirable to eliminate all such explicit reference to wall distance.

Another limitation may be seen by considering the special case of a uniform mean flow, with both velocity derivatives vanishing. In that case the length scale, as defined in the model, becomes indeterminate. In a sense, this is a limitation of the model. However, it is correct in telling us that under these circumstances, the applicable length scale is determined by other factors. This does not seem to be a serious limitation for a model intended primarily to deal with shear flows.

Notice that length scale at a point is defined not in terms of the velocity derivatives at that point, but rather in terms of average values over a finite region enclosing the point. This eliminates the difficulties which would otherwise arise, for example, at a point of inflection in the velocity profile.

Question:

In many practical problems, the characteristics of turbulence in pressure gradients is important. Do you have any information about that, or any plans to look at that aspect?

Answer:

The question of pressure gradients is important in connection with boundary layers, especially in regard to separation. This application would represent an excellent test of the heuristic model. Yes, we are definitely interested in further applications of the heuristic model to this and other cases, assuming that the necessary support is obtained for this purpose.

However, our greatest activity right now is with the two-dimensional integration of the unsteady equations. In the channel flow problem, the pressure is not decisive in the same way as for a boundary layer. In fact, it does not even occur explicitly in the final equations. The purpose of this work is simply to learn more about the basic mechanism of turbulence in itself. If we make progress in this, the applications to boundary layers, and to questions of pressure gradients and the like, will follow as a matter of course.

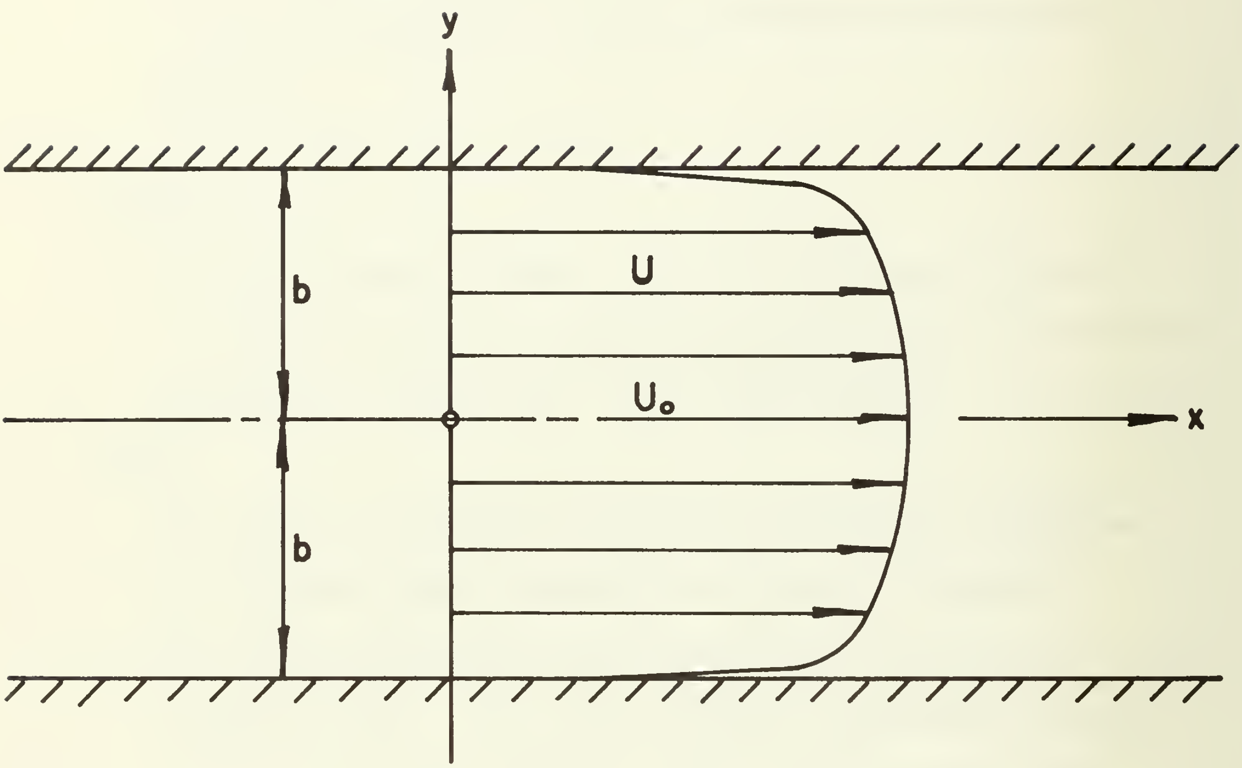


FIGURE 3-1 Flow in a Two-Dimensional Channel

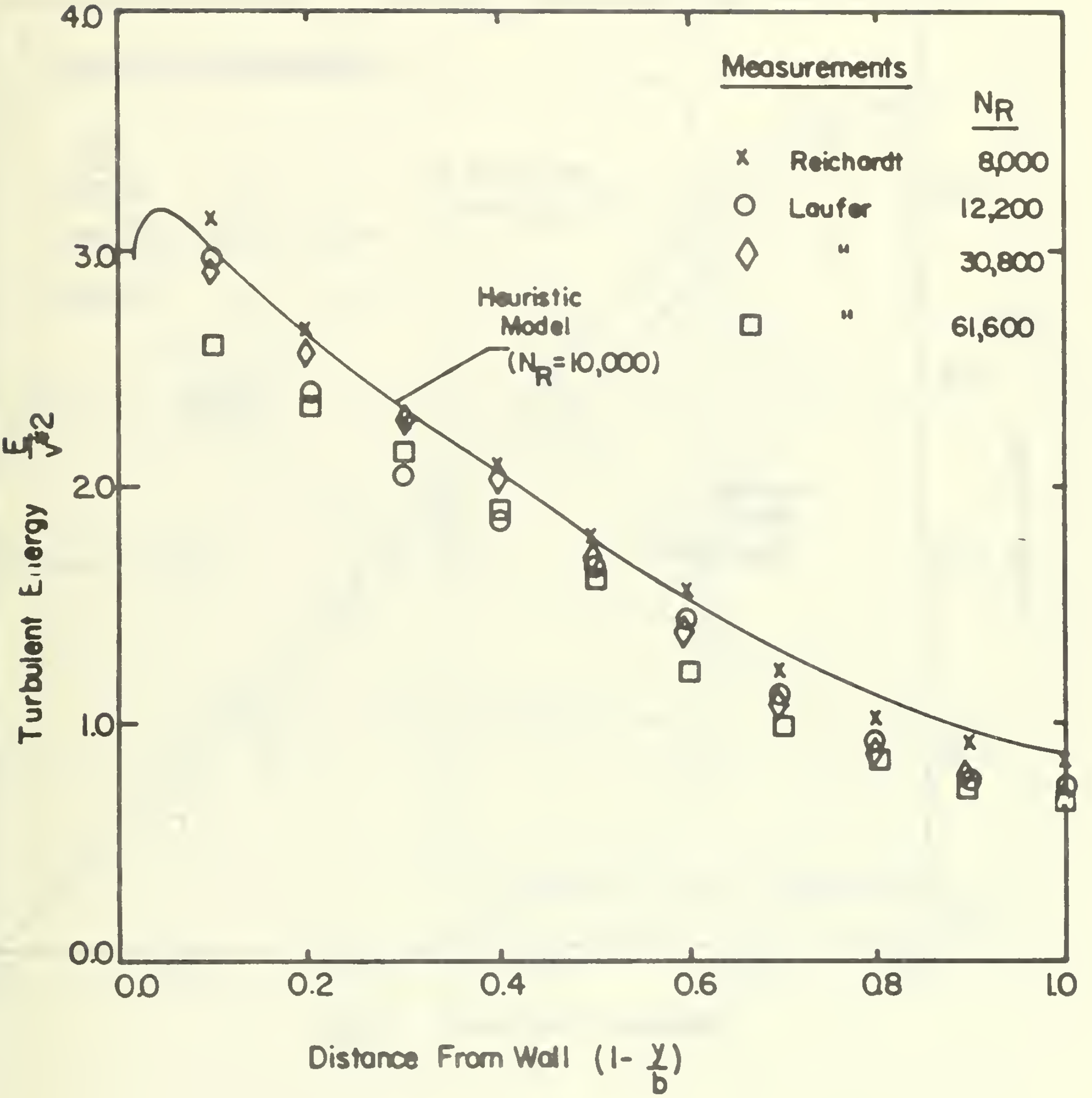


FIGURE 3-2 Turbulent Energy Distribution in a Two-Dimensional Channel

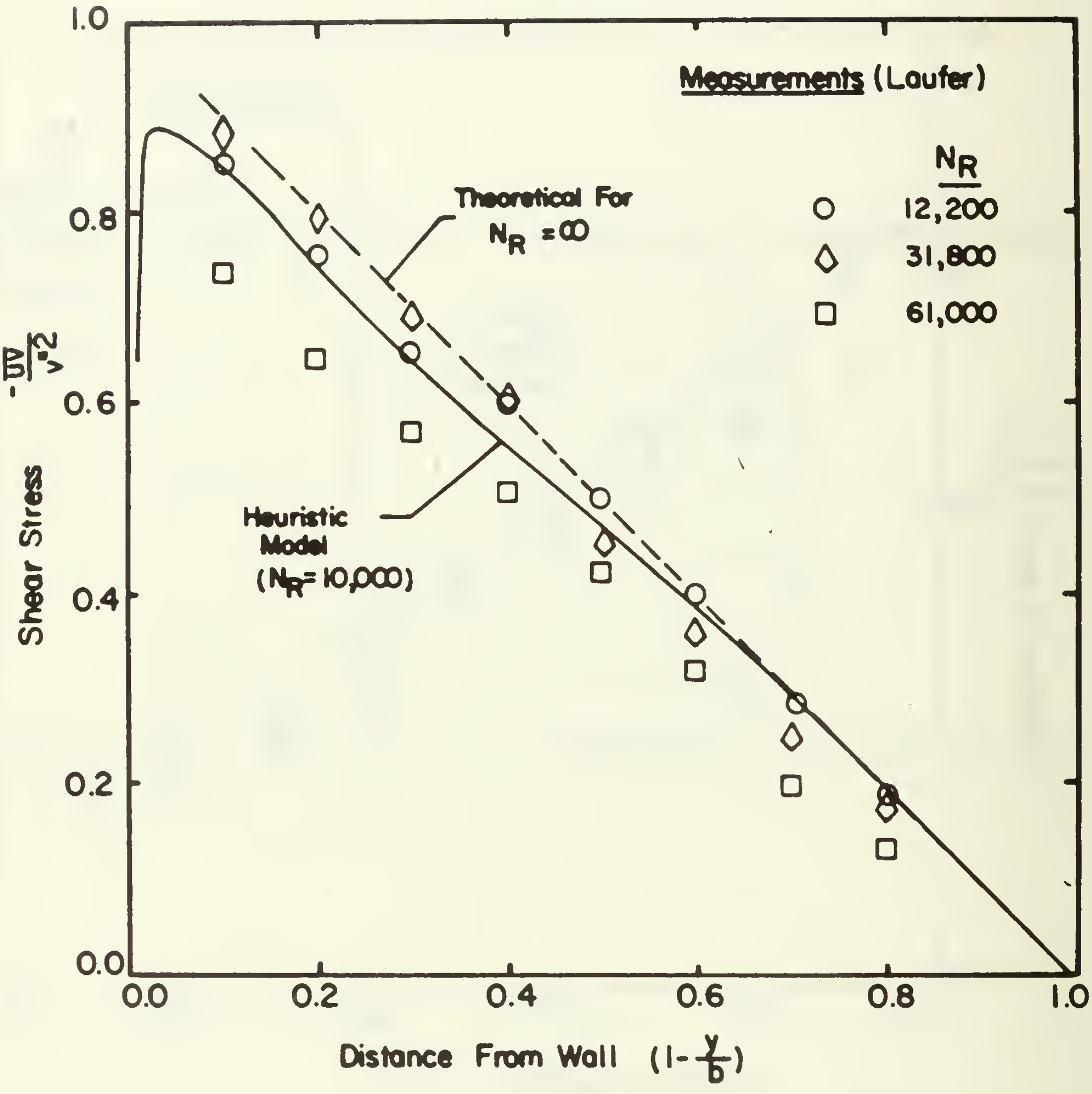


FIGURE 3-3 Reynolds Shear Stress Distribution in a Two-Dimensional Channel

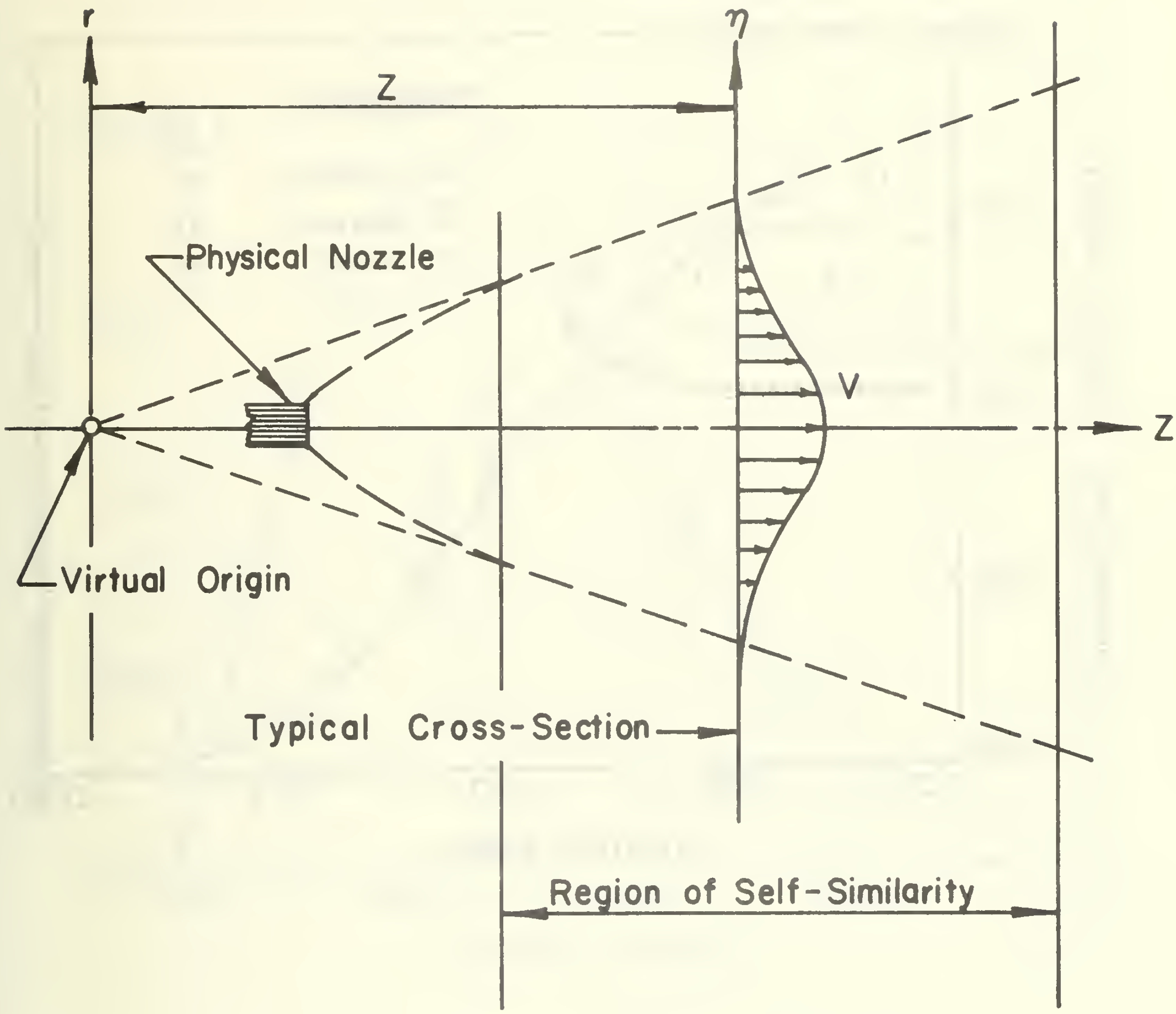


FIGURE 3-4 The Axi-Symmetric Jet

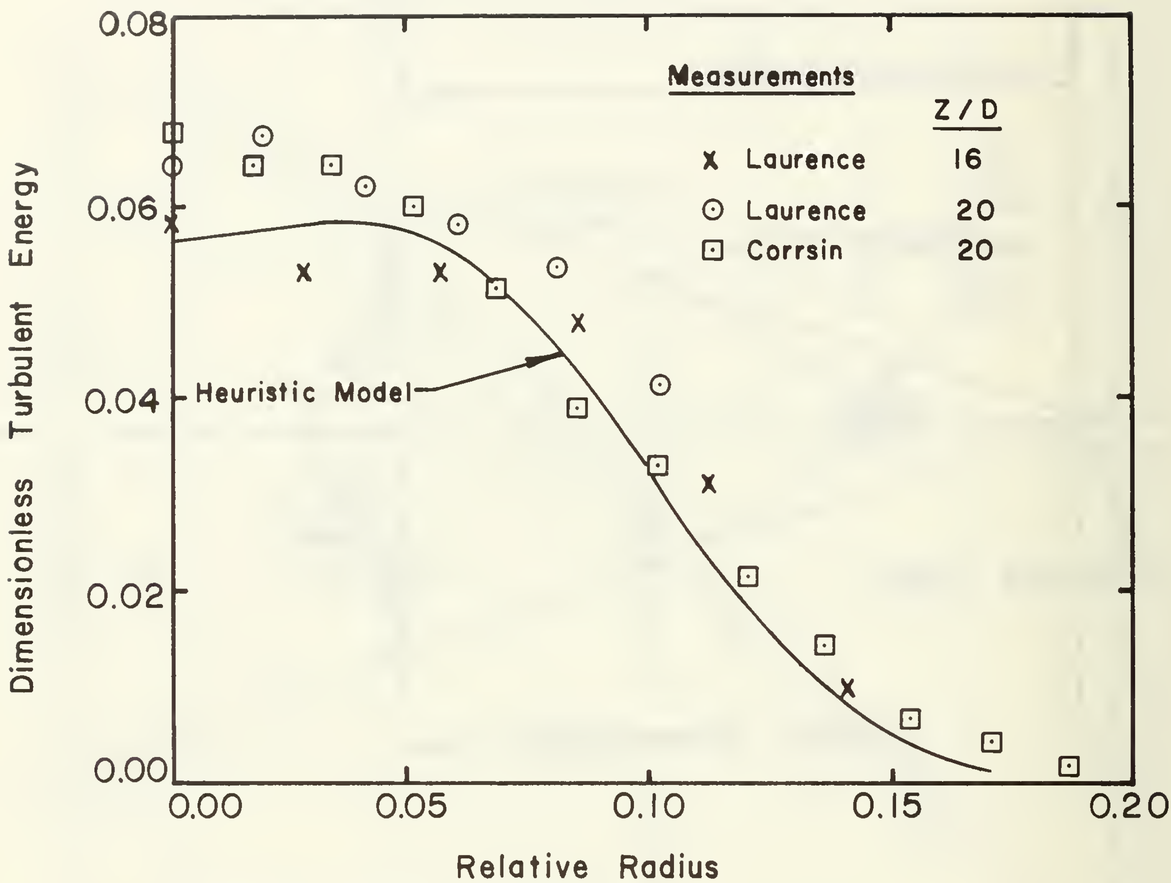


FIGURE 3-5 Turbulent Energy Distribution in a Circular Jet

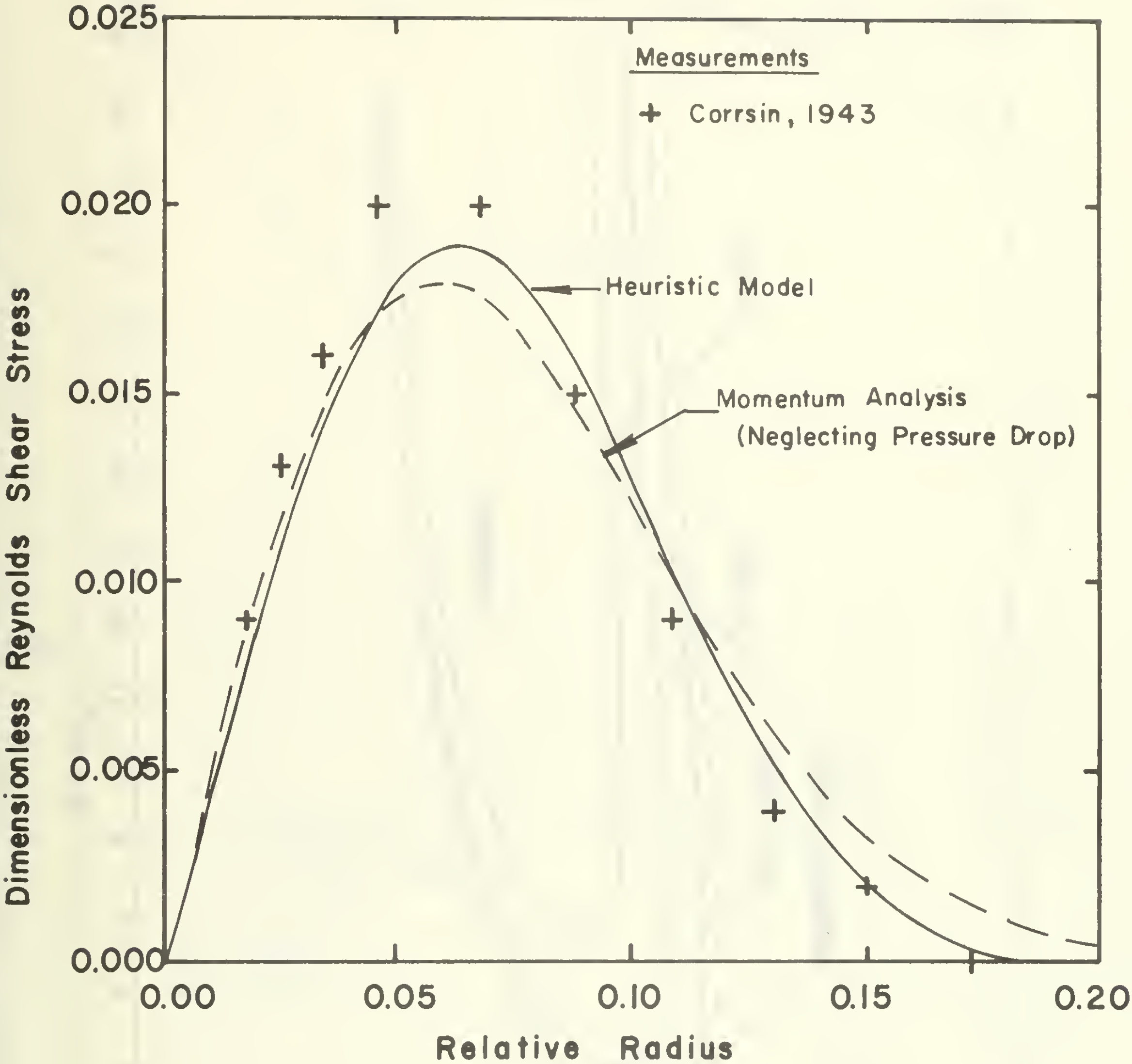


FIGURE 3-6 Reynolds Shear Stress Distribution in a Circular Jet

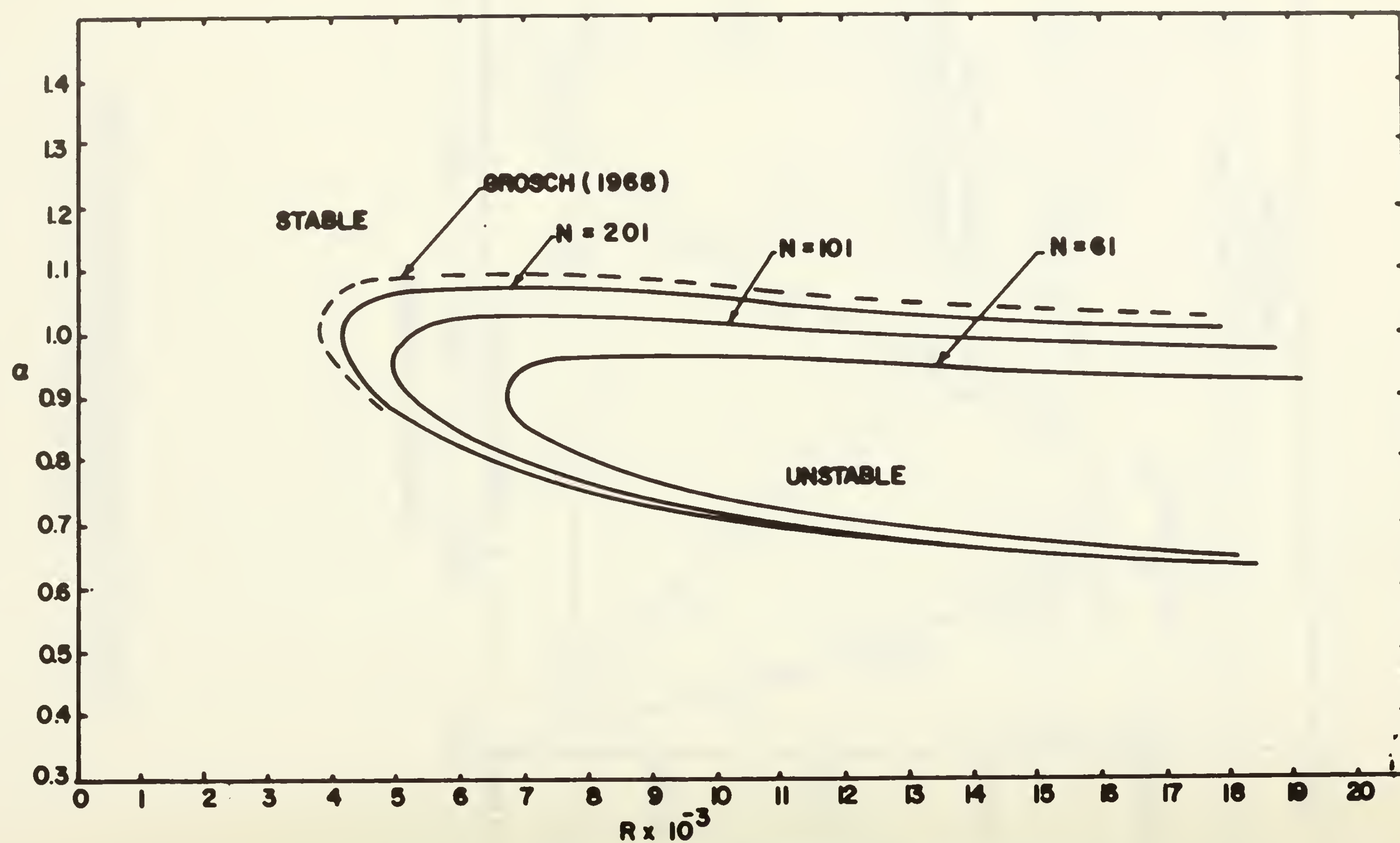


FIGURE 4.1-1 Effect of Finite Difference Mesh Size on Curve of Neutral Stability for Plane Poiseuille Flow

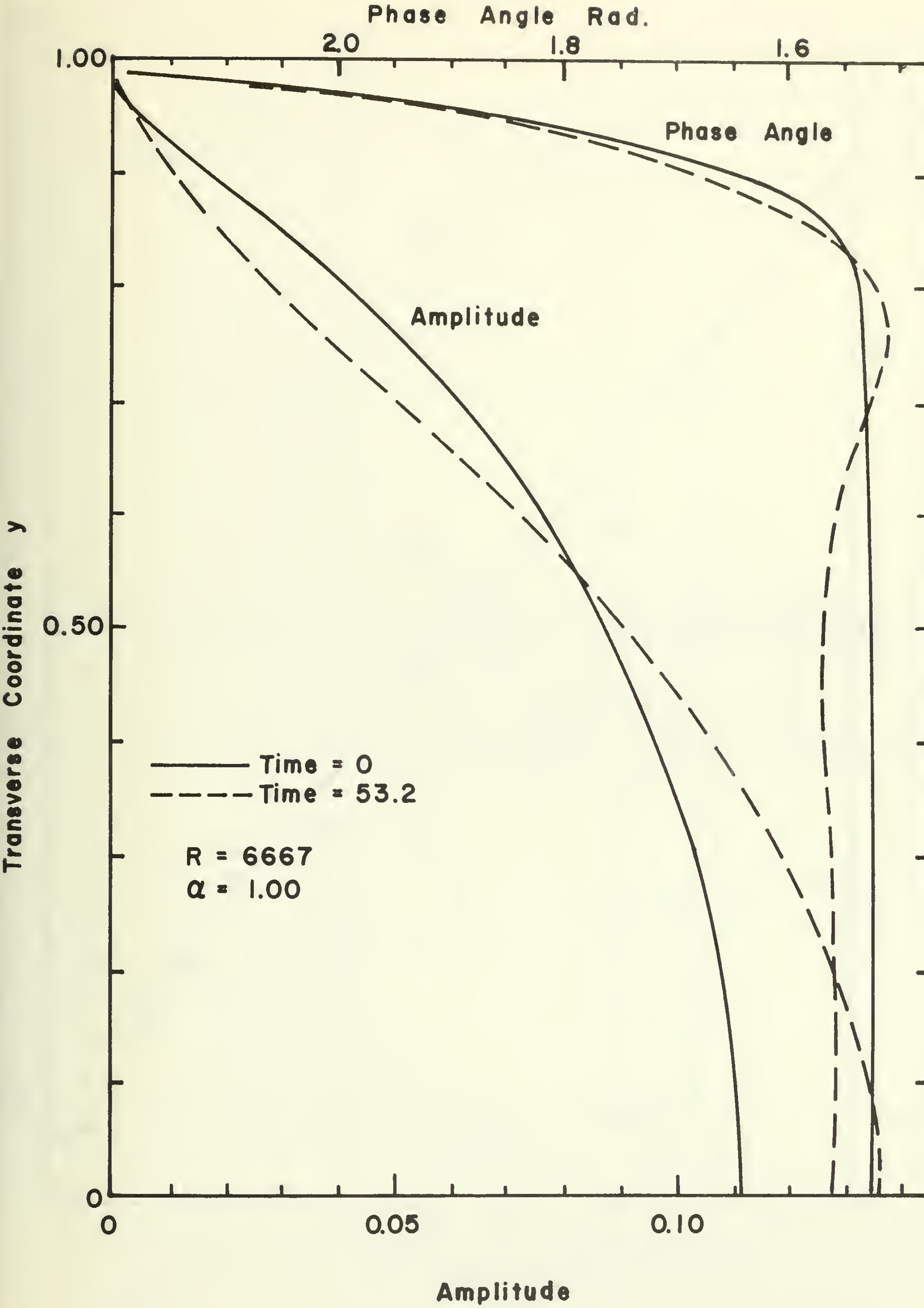


FIGURE 4.2-1 Change in Amplitude and Phase Angle for First Mode

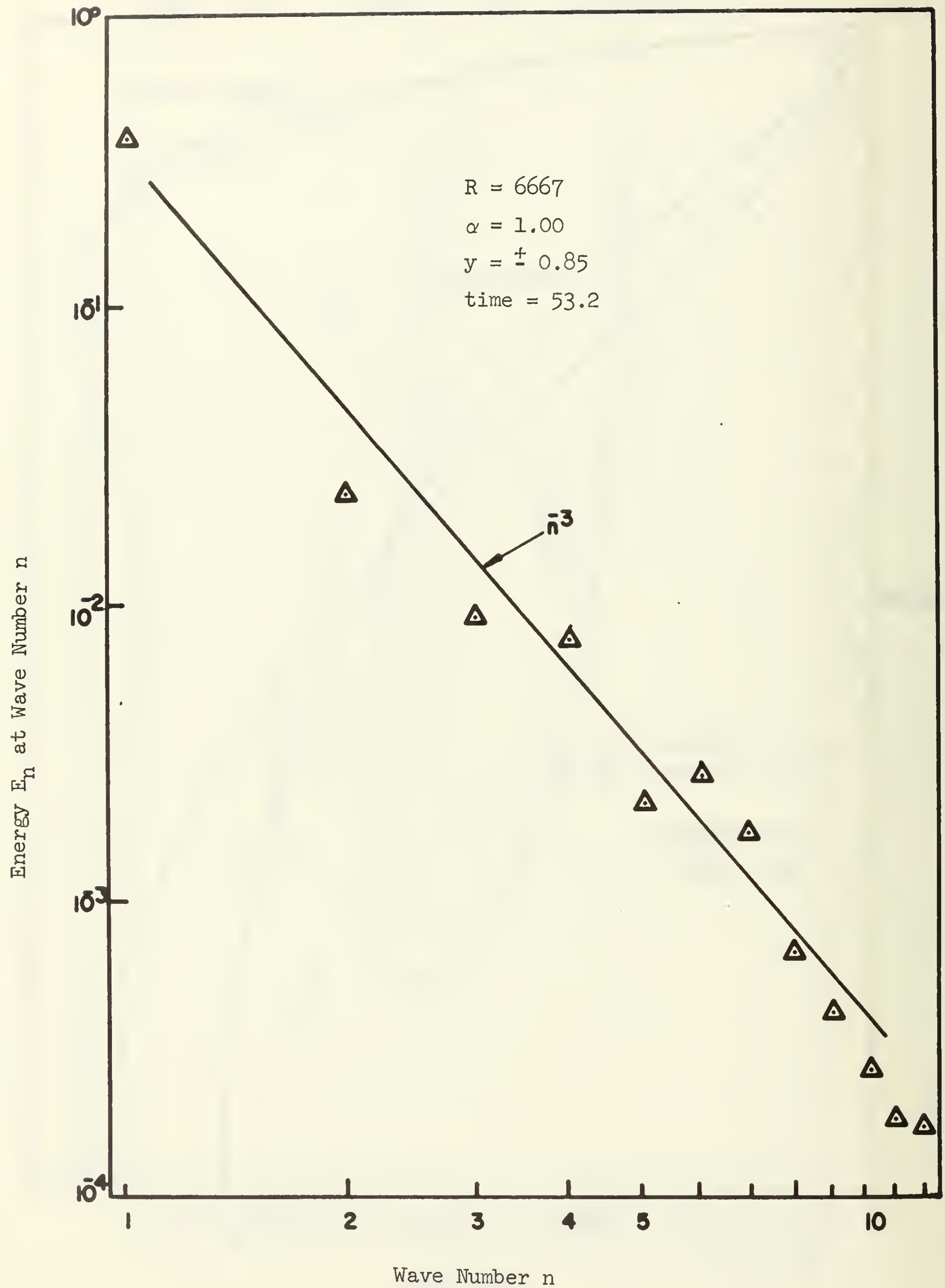


FIGURE 4.2-2 Typical Turbulent Energy Spectrum

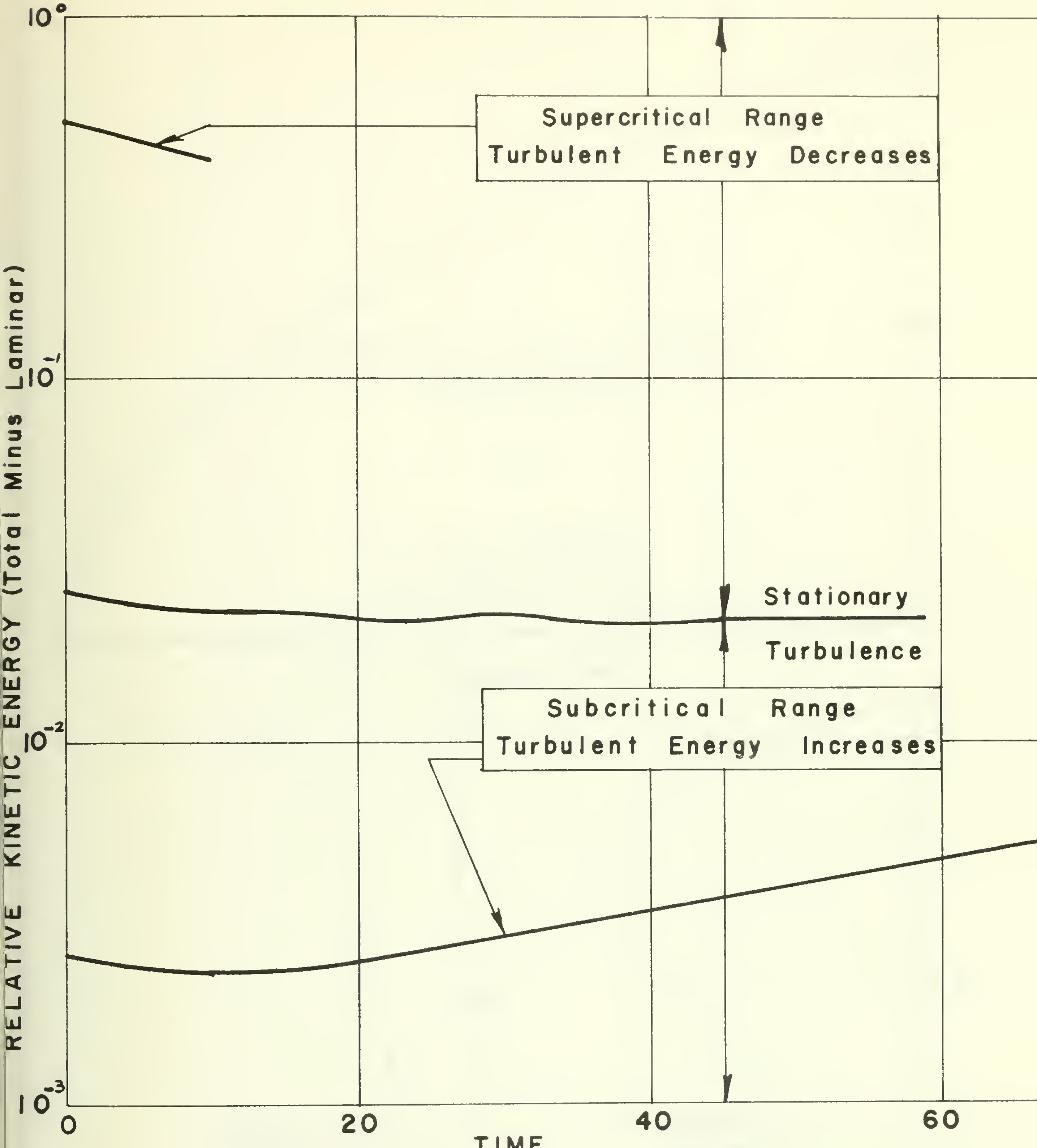
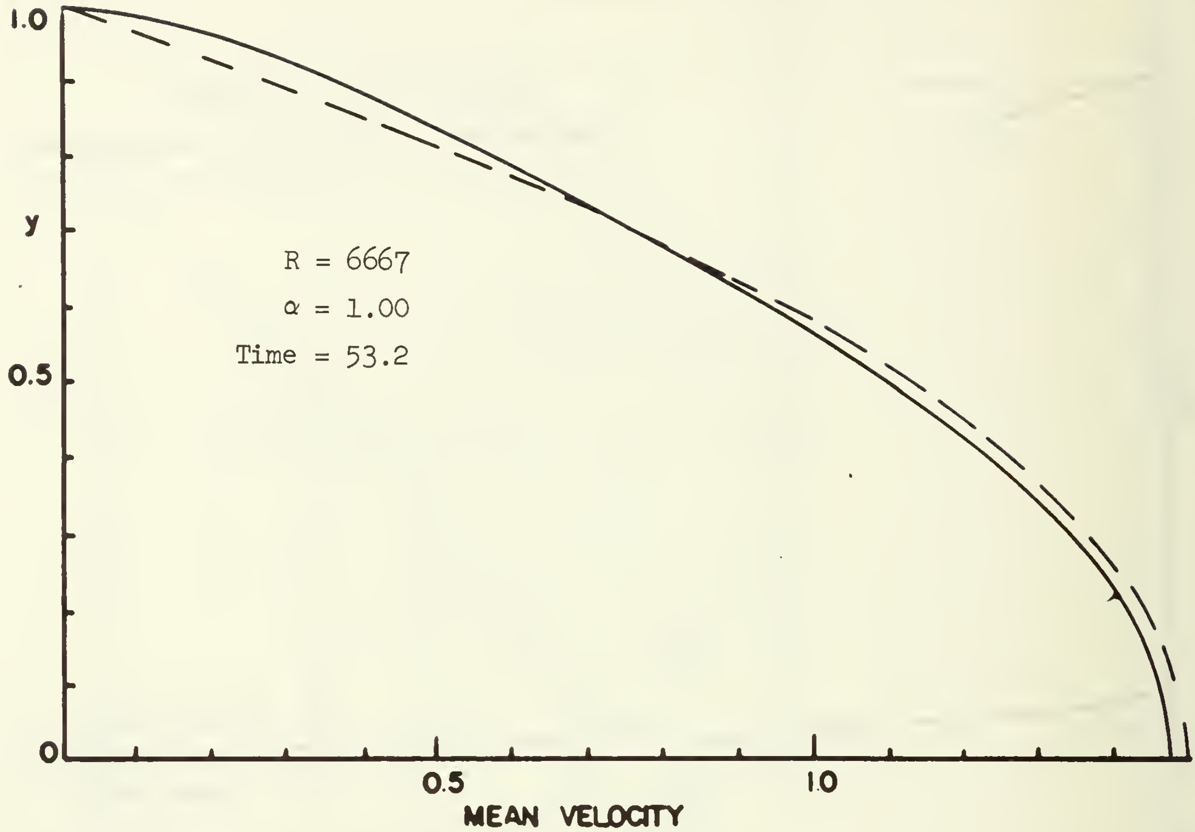
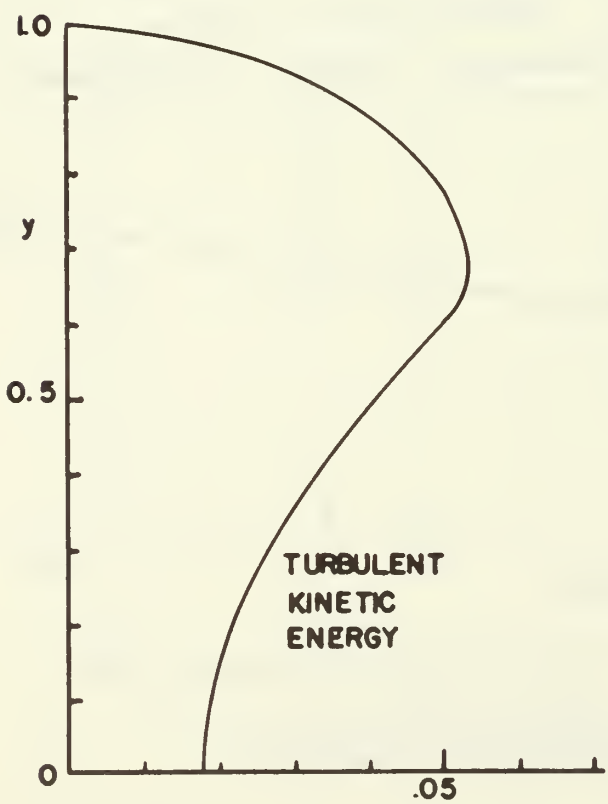


FIGURE 4.2-3 Evolution of Turbulent Energy for Three Different Initial Amplitudes





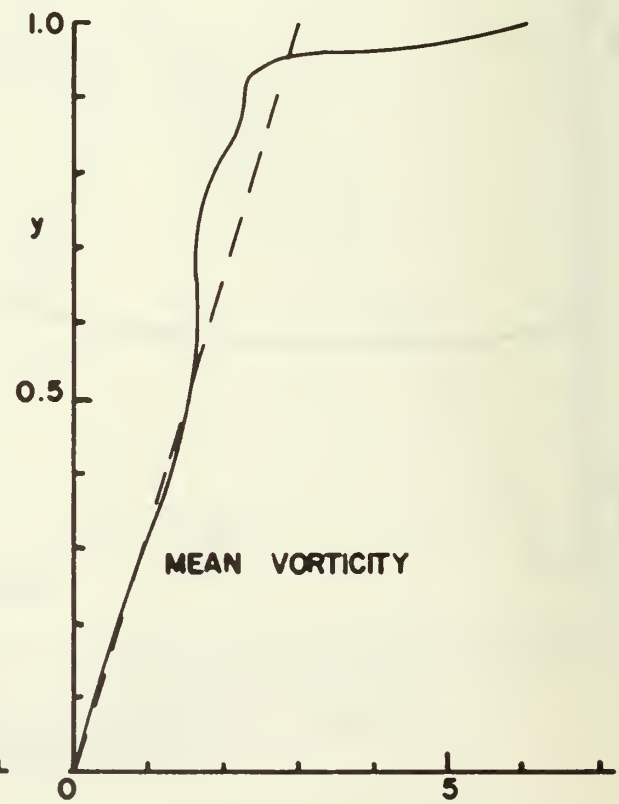


FIGURE 4.2-4 Typical Distribution of Mean Velocity, Vorticity and Turbulent Energy

A REVIEW OF THE HISTORY OF BOUNDARY LAYER CALCULATION METHODS AND THE PRESENT STATE OF THE ART

by

Dr. K. D. Papailiou
Assistant Professor
Naval Postgraduate School

LIST OF FIGURES

INTRODUCTION

An attempt is made below to review the history of boundary layer calculation methods as well as the present state of knowledge. This review is by no means claimed to be complete. It is limited by my own experience in boundary layers. Consequently, turbulent boundary layers and their applications in turbomachines are particularly stressed.

Throughout the text I tried to cite references where one would not only find a more detailed discussion of the subject mentioned but also all the pertinent references on it.

A REVIEW OF THE HISTORY OF BOUNDARY LAYER CALCULATION METHODS AND THE PRESENT STATE OF THE ART

1. Statement of the Problem for Laminar and Turbulent Boundary Layers. Essential Differences.

Once it is accepted that the relations between stress and rate of strain and between heat flux and temperature gradient are linear, one arrives at the Navier-Stokes equations, the most general equations we have at our disposal for the moment relating the macroscopic quantities that describe the flow of a fluid.

This assumption can be traced back to the theory of Gaskinetics.[1,2,3,4,5,6]

The numerical solution of the Navier-Stokes equations is a formidable task; and even with today's modern computing equipment, solutions have been obtained only for very simple boundary values and for Reynolds numbers of the order of one thousand. It suffices to say that attacking numerically the Navier-Stokes equations means entering the 10^{15}-operations era[7]. For a mean time of 1 μsec per operation, which is what can be achieved with today's computing equipment, the time needed for computation is approximately 30 years.

The simplified flow model used in order to avoid coping with the complete Navier-Stokes equations is the one which divides the flow into two interacting regions:

(a) The inviscid region where viscous effects are considered absent

(b) The boundary layer and wake region, where viscous effects must be taken into account. This region reduces to a thin layer adjacent to the solid surface for high Reynolds numbers based on overall quantities. For this case, which is of importance to the aerodynamicist, the Navier-Stokes equations reduce to the boundary layer equations (Prandtl's assumptions). Experiment has proved that laminar flows are stable for low Reynolds numbers based on local quantities, but that they become unstable for higher ones. A new pattern of flow appears called turbulent, which is characterized by the highly irregular motion of fluid particle anglomerations. The point from whereon disturbances tend to amplify can be calculated using the laminar boundary layer (LBL) equations, at least up to a Mach number of approximately 3, up to which Tollmien-Schlichting's instability mode is the stronger one[8]. No information can be obtained either from the LBL equations or from the Navier-Stokes equations about the origin and nature of the turbulent pattern. It is customary to apply the statistical mechanics approach to obtain tractable equations from the Navier-Stokes equations in view of the fact that the highly irregular turbulent motion has a definite pattern of a random nature. However, certain features have to be possessed by a system in order to be able to apply the statistical mechanics approach to it. For the system of the turbulence, most of these features are not satisfactorily solved including the Ergodic Theorem, which establishes the equivalence between statistical and time averages.[6]

By decomposing the flow properties into a time mean value and a fluctuating one, introducing this expression into the Navier-Stokes equations, taking averages, and then applying the boundary layer assumptions, we come to the turbulent boundary layer equations. These equations include as additional unknowns averaged combinations of fluctuation terms. They are, however, more tractable than the unsteady Navier-Stokes equations and can be treated numerically if additional hypotheses are formulated for the turbulence properties, which are the additional unknowns.

Although the laminar and turbulent BL equations are almost the same in form, the transport properties of the two are essentially different. The momentum exchange between adjacent layers, the direct outcome of which is the stresses, is realized on the molecular scale for the LBL's (transport coefficients are properties of the fluid) and on the particle anglomeration scale for TBL's (transport coefficients are properties of the flow).

The laminar boundary layer equations are parabolic in nature so that only upstream history influences what happens at a station. This upstream history is included in the equations' structure. For the TBL's there is an additional part of upstream history which has to be obtained from experiment. This concerns the turbulence properties and the assumptions which are postulated for them. The nature of the final equations governing the TBL will depend on these assumptions. Up to now they have been either parabolic or hyperbolic in nature so that again only upstream history influences what happens at a station.

The formulation of the problem for the turbulent properties is an essential one if one takes into account that the intensity of the transport processes is one order of magnitude higher for TBL's than for LBL's. It necessitates an additional equation of the same nature of the BL equations to be introduced if the turbulent properties are to be considered as flow properties.

In conclusion we can summarize the following points:

(a) The question of the validity of the Navier-Stokes equations is left open, although objections have only been raised for their application in low densities and high temperatures.

(b) For LBL's the problem is mathematically well-formulated; and if difficulties are encountered, they are either of numerical order or due to the fact that the BL approximations break down.

(c) For TBL's the problem is mathematically well-formulated if additional equations are furnished for the turbulent properties. These equations are necessarily semi-empirical since the nature of turbulence remains unknown, hence the uncertainty of the solution of the turbulent problem.

2. Early Approaches and Solutions for the Two-Dimensional Incompressible and Compressible BL.

The early days of methods for the numerical solution of the BL equations are marked by several facts.

(a) By the lack of fast computing equipment. This excluded any attempt to solve the partial differential equations for both laminar and turbulent BL's. The partial differential equation was replaced by a number of integral moments of it (usually two). At the very beginning the momentum integral equation was used, coupled with an empirical relation which was sometimes an algebraic one. Later the empirical equation was substituted by another integral moment of the original partial differential equation.

As it is well known, a partial differential equation is equivalent to an infinite number of ordinary ones which are the integral moments of it. Consequently, the above-mentioned formulation not only disregarded this point (on which we shall comment later) but also necessitated the formulation of additional assumptions concerning the additional unknowns resulting from using only a part of the available information. The integral moments used being two, a one parameter velocity profile family was postulated. This assumption is a sufficient one in order to arrive at the solution.

(b) By the fact that our knowledge of turbulence was very limited. This fact obliged the investigators to stick to solutions employing integral moments only, as it was found much preferable to postulate assumptions for overall turbulent quantities than to try to describe in detail the turbulent properties at each y-position inside the TBL. This fact delayed by a decade the appearance of methods solving the partial differential equation for TBL's.

Additionally, because of both facts (a) and (b), the assumptions concerning the turbulent quantities were kept very simple. In fact up to the early 60's it was usual practice to relate explicitely or implicitely the velocity profile gradient to the shearing stress or, more generally, to postulate the assumption of local similarity. In order to specify this assumption,one must specify first the equilibrium boundary layer.[11,12,13] An equilibrium BL is the one for which the acting forces (inertia, pressure and friction) are kept in the same ratio throughout its development. This layer has a constant past history. Additionally, the diffusion terms are zero, so that the turbulent energy produced is dissipated at the same station. Such a layer is free from its past history.

The assumption of local similarity postulates that any station of an arbitrary boundary layer corresponds to a station of an equilibrium BL (thus the past history as far as turbulent properties are concerned is neglected). This assumption implies that the turbulent properties are related to local quantities.

(c) By the lack of accurate experimental results. There was very recently made an evaluation of the available experimental data.[9] Many of the old experiments had to be rejected on the grounds of uncontrollable three-dimensional flows present. It was on these experiments that the old methods were based.

During the early days of BL-calculation methods, attention was payed mainly to the two-dimensional, incompressible BL. The calculation of the compressible, laminar boundary layer was done through a transformation to an equivalent incompressible one (Stewartson-Illingworth transformation[14]) for adiabatic walls or by approximate methods subject to further restrictions[15,16] (Prandtl number equal to unity and a linear variation of viscosity with temperature) for problems with heat transfer. A general outline of the approximate methods can be found in ref. 17.

For compressible TBL's the transformation technique was used most times because of the complexity of the problem.

This first period in BL calculation methods was ended for LBL's by the advent of modern computing equipment, which gave the opportunity to avoid unnecessary approximations. (Late 50's). For TBL's it ended with the work of Rotta[12] and Thomson[18] who made evident the insufficiency of the methods available up to then for the calculation of TBL's.

3. Field and Integral Methods. Different Schools.

It is customary to classify today the calculation methods for boundary layers into field or differential and integral methods. Field methods are those which employ the partial differential equation, solving an equivalent finite-difference scheme. Integral methods are those which solve a system of integral moments of the BL equations.

Three main schools were developed in the 50's inside the frame of integral methods. They were named after the specific integral equation they used to couple the von Karman integral equation with: (a) the energy school, (b) the moment of momentum school, (c) the entrainment school.

These three schools retained their character during the 60's even when additional equations were used to take into account the past history of the boundary layer.

4. Comparison of Experimental Data with Theoretical Prediction for TBL's.

Rotta[12] in 1962 did the first serious attempt to compare theoretical predictions with available experimental data by noting that any integral method could develop the auxiliary equation in the form:

$$L\,\theta\,\frac{dP}{dx} = -\,M\,(P,\ Re_\theta)\,\frac{\theta}{U_\infty}\,\frac{dU_\infty}{dx} - N\,(P,\ Re_\theta)$$

where:

L assumes the value 0 or 1

P is the form factor used

θ is the momentum thickness

U_∞ is the velocity at the edge of the boundary layer

$Re_\theta = \frac{U_\infty \theta}{\nu}$

$M(P, Re_\theta)$, $N(P, Re_\theta)$ are functions of P and Re_θ

Rotta compared the functions M and N with the available experimental results for equilibrium TBL's ($\frac{dP}{dx} = 0$). The results are presented in Figure 1. One can observe the insufficiency of the old methods. Thomson's results confirmed Rotta's conclusions and showed further that the best available method until then was the entrainment method of Head.

5. The Question of Equivalence Between the Partial Differential Equation and the Number of Integral Moments (The Work of Bethel).

As we have already remarked, a partial differential equation is equivalent to an infinite number of ordinary differential equations which are the integral moments of it. The mathematical formulation of an approximate solution using a finite number of moments has been given by Dorodnitsyn[19]. Numerical solutions using up to ten moments have been obtained by Bethel[20] for laminar, incompressible, two-dimensional BL's. As the number of integral moments increases, the conditions imposed on the velocity profiles in order to establish a mathematically well-formulated problem are relaxed. A typical set of results is presented in Figure 3, where the approximate solution is compared to the exact one.

The conclusions of Bethel's work can be summarized as follows:

(a) The solution is rather insensitive to the type of velocity profile used, a fact which has also been recognized for turbulent BL's[9] if the shear stress profile is "unhooked" from the velocity profile gradient.

(b) The solution converges rapidly to the exact one as the number of equations increases, a fact which justifies the use of a limited number of integral moments in the place of the partial differential equation without destroying the effects of the past history in that respect. One could argue that the same thing is true also for TBL's in view of the success with which a two-dimensional TBL can be calculated today using three differential equations.

6. Modern Formulation Methods for the Solution of Turbulent Boundary Layers. History Effects.

Once the insufficiency of the old methods was made clear by the work of Rotta and Thomson, active research was undertaken and new methods were developed, both field and integral, to solve the problem mainly in the two-dimensional incompressible and compressible cases.

It is rather significant that in the 1968 Turbulent Boundary Layer Conference at Stanford[7,9] most of the methods presented were developed after the 60's and the few old ones were recast and redeveloped in a new form. The new methods developed differed essentially from the old ones in that history effects were taken into account as far as the turbulence properties are concerned. The shear stress profile was "unhooked" from the velocity profile; and another equation was introduced to account for the fact that the turbulent properties didn't adjust themselves instantly to local conditions, but their values at a certain station depended also on what has happened upstream. For the integral methods this new equation is usually an empirical ordinary differential equation, called the lag equation, which essentially describes the departure from equilibrium conditions (see for instance refs. 21, 22, 23, 24). For the field methods this new equation is either the turbulent kinetic energy equation[25,26] or an eddy viscosity or mixing length formula[27]. Although the field methods are not more accurate than the best integral methods,[7] it must be recognized that the use of the turbulent energy equation, along with the assumption that the turbulence properties are related between the two, makes more sense physically than the introduction of the lag equation.

7. The State of the Art after the Stanford Conference.

An important step towards clarification of today's knowledge was acccomplished through Stanford's conference on computation of turbulent boundary layers in 1968[7,9]. Although several positive steps in clarifying where we stand were made and the advances of the last decade's research were brought to light, today's insufficient knowledge of the fundamentals of turbulence was made clear.

The available experimental data were examined and many of the old experiments were discarded because they were found to contain strong three-dimensional effects. The lack of adequate experimental data even for two-dimensional incompressible TBL's was realized, and the non-existence of experimental data in the domain of compressible and three-dimensional TBL's was stressed.

Many of the available methods were found quite accurate for the prediction of two-dimensional boundary layers in spite of the fact that a better understanding of the physical aspects of the problem is still lacking. Although Stanford's conference concerned two-dimensional, incompressible boundary layers, comments were made in other domains and they will be reviewed in the following section. Before going any further, a summary of what has been said up to now will be made along with certain comments in order to lay the foundation for the discussion to follow in the next section.

The first difficulty that hindered the development of both laminar and turbulent BL calculation methods was of numerical order. An idea of the numerical problems involved can be formed by looking at Figure 4, which has been reproduced from ref. 7. The advent of modern computing equipment has helped greatly in displacing the barrier, but not in eliminating it completely. Simplified models and consequently comparison with experiment are necessary in certain cases where exact solutions don't exist, even for laminar boundary layers where the problem is purely a numerical one.

In addition one has to take into consideration the fact that BL calculations constitute only a part in any design procedure so that involved and lengthy calculations, even if they are accurate, are often rejected on these grounds. Historically speaking we have seen that this difficulty hindered the development of two-dimensional BL calculation methods up to the late fifties.

A second difficulty was that related to the accurate representation of the past history of the boundary layer. In this respect we have seen that, while for LBL's the problem of representing the past history is posed only when the partial differential equations are replaced by a number of integral moments (which are more easy to compute), for TBL's we have to introduce additionally the part of the past history agents of which are the turbulent quantities. This has to be done on an experimental basis, as the turbulent quantities are the additional unknowns of our problem.

Last decade's research is marked by the development of modern computing equipment which oriented research in the development of calculation methods. With today's fast computing machines it is possible to use more sophisticated models, which are also more complicated, either to attack the problem of representation of the past history of the BL or to attack problems for which a higher approximation to the Navier-Stokes equations is needed (breakdown of BL equations) or a combination of the two.

As there are generally no exact solutions in these domains, these models are the outcome of understanding of the flow behavior on the basis of the available experimental evidence. As it was realized in the late sixties, the available experimental evidence was meager and insufficient for the understanding of all the important aspects of the problem and consequently for the formulation of realistic models. This fact consititutes the third difficulty present which hinders the development of successful calculation methods.

Speaking from a fundamental point of view, we have to admit that these three difficulties are far from being solved. However, accepting the "black box" view of the problem, as imposed by the current needs of practical applications and looking at things from an engineering point of view, we have to acknowledge the recent advances realized during the past decade. They helped to identify and pose a certain number of important problems, the influence of which remained completely unknown, and to "unhook" the scientists from the "academic" problems and focus their attention to the "real-life" problems.

There exists today a formulation for practically all identified problems. If one considers the fact that however crude this formulation is, it presents an approximation of varying degree of accuracy to the real problem; then one may come to the conclusion that if this formulation is used, having always in mind its limitations, it constitutes a very useful tool for the engineer. As an example for what can be achieved with today's knowledge, we present the test results (Figure 5 from ref. 38) for a 45° turning compressor blade[35] calculated theoretically on the basis of a simple boundary layer calculation method[39] as compared with the experimental results with the NACA series blades.

8. What Can and What Cannot be Calculated with Today's Knowledge? Problems to be Solved.

We shall confine ourselves to problems concerning mainly the field of turbomachines. Our discussion will concern flows up to a Mach number of approximately three.

8.1 Two-Dimensional Boundary Layers.

Incompressible two-dimensional laminar boundary layers can be calculated fairly easily with today's computational means using either field or integral methods. A discussion of the methods available and their accuracy can be found in references 28,29,30,32. For compressible laminar boundary layers also the problem seems well in hand,[31,32] and the semi-empirical formulae for heat transfer and other properties seem accurate enough for engineering applications. The instability point up to a Mach number of 2.5, up to which Tollmien-Schlichting instability is the principal mode[8], can be calculated fairly accurately (for calculation see also ref. 33). The calculation of the transition region, however, remains unsatisfactory although several correlations exists.[8,34,35,36,42] They all rely heavily on the "know-how" of the user.

The calculation of the incompressible turbulent boundary layers is now well in hand.[7,9] Successful correlations exist for the turbulence quantities. Figures 6 and 7 give an idea of the accuracy with which a two-dimensional turbulent boundary layer can be calculated (Bradshaw's method, ref. 25, and the new entrainment method of Head, ref. 37).

It might be of interest to note that the two-dimensional turbulent boundary layers encountered in turbomachinery are usually monotonically diffusing. This class of boundary layers is rather well behaved and remains close to equilibrium conditions (see Figure 8 taken from ref. 23). This fact allows one to use simple theories based on local similarity.

As stated in reference 40 there is now sufficient experimental evidence (see also refs. 7,30,41) to support Morkovin's hypothesis which states that the structure of turbulence doesn't change with Mach number much as long as the Mach number based on a typical fluctuation velocity is small. This fact enables us to calculate compressible turbulent boundary layers up to Mach numbers 2.5 or 3, that is within the field of applications to turbomachinery, using the correlations developed for incompressible flow.

However, the fact must be stressed[9] that there is a lack of sufficient experimental data to test a compressible turbulent boundary layer calculation method.

8.2 Three-Dimensional Boundary Layers.

When discussing three-dimensional boundary layers, one has to realize that even for LBL's where the problem is well formulated, important numerical difficulties arise. Formulations of the problem exist in both field and integral methods.[43,44,45,46,47,48] Raetz's formulation is the most widely acceptable. However, it remains still questionable until more experimental data for testing it are available.

Calculation methods for the case of the 3D turbulent boundary layers exist in both the integral and field methods.[49,50,51,52,53,7,40,45] In addition to the numerical difficulties mentioned above, the question of "what is the direction of the shear stress vector" has to be answered. The different methods solve this question in a different manner. However, available experimental evidence is contradictory, at least for flows with strong crossflow. No conclusions can be drawn, and the need for additional careful experiments in 3D flows prevails. With the present state of knowledge only 3D TBL's with small cross flows can be calculated.

In the class of the 3D boundary layers, the boundary layer in a corner has to be considered, although it is quite different in nature. Although formulations exist[54,55,56] this problem, which constitutes the first important step for the solution and understanding of the secondary loss problem, is far from being solved.

8.3 Separation and Separation Criteria.

The importance of being able not only to predict separation but also to penetrate and calculate what happens inside the separated flow region is now well established. Even in the simple case of a two-dimensional compressor cascade, one knows that many times optimum performance is achieved with partial separation. On the other hand one has to realize that objections may arise in using the boundary layer equations through the separated region and inside the separated layer on the grounds that;

(a) the boundary layer approximations fail near separation,

(b) the flow downstream of the separation region is quite different in nature from the one upstream. For instance, while upstream of the separation region for laminar subsonic 2D layers the flow is parabolic in nature, it becomes elliptic downstream.

(c) the interaction between boundary layer and external inviscid flow must be considered, when the external pressure field is changed due to the presence of the separation region as is many times the case, and

(d) separation itself is a three-dimensional and in many cases unsteady phenomenon even for a two-dimensional turbulent boundary layer.

Discussion on the nature of separation can be found in references 7,12,28,57,58,59,60,61,62,63,64,65,66. No definite conclusion can be drawn up to now as opinions diverge even on the definition of separation.[7] Separation criteria were developed very early[67] to cover engineering needs and they have been used with varying success depending on their degree of sophistication. In fact, if the pressure distribution is not radically changed with the appearance of separation, as it happens in many cases in turbomachines, once the inviscid pressure distribution is calculated, the separation region can be predicted with a certain degree of accuracy. Although each boundary layer prediction method has its own separation criterion built in, the most widely applicable ones are those proposed by Buri[67], Maskell[68], Stratford[69] and the ones using a value of H_{12}, the momentum form factor, equal to 2.2 approximately or a zero value for the wall shear stress. It is worth noting that Sandborn[66] recently proposed a new criterion for separation prediction taking into account the unsteady nature of the phenomenon.

Engineers in need of loading criteria for design purposes have developed some of the existing local separation criteria into criteria containing overall quantities as the ones presented in refs. 70 and 71. This oversimplification restricts even more the validity of the separation criteria to flows created by the particular designs, which were used as models for the development, and should be applied with care.

8.4 The Influence of Secondary Effects on the Boundary Layer Developme

There is a certain number of problems the importance of which has been recognized recently and which have not yet been fully investigated. They concern "real-life" effects that have to be considered in a turbulent boundary layer calculation. The importance of these effects is demonstrated in Table I, taken from ref. 40. They are discussed below.

(a) The effect of flow convergence on turbulence is not yet fully understood[7]. The current approximation accomplished consists in modifying the continuity equation accordingly[35,72]. An idea of the order of magnitude of the effect can be demonstrated by the following example. For a 45° turning compressor blading tested with an aspect ratio of unity, the corner vortices introduced a flow convergence that increased the "two-dimensional" losses by a hundred per cent. Flow convergence modifies essentially the momentum thickness while it has little influence on the momentum form factor

(b) The effect of small Reynolds Numbers

The work of Coles[73] and the visualization studies of Patel and Head[74,52] indicate that Reynolds number effects must be taken into account for $Re_\theta < 5000$, as they affect the structure of turbulence. The turbulent boundary layer in turbomachines starts with values around 200 and it rarely attains as high values as 10,000.

(c) The effect of streamline curvature.

Longitudinal surface curvature has very little influence on laminar boundary layers[75]. However, it strongly influences turbulence. This effect was recognized and incorporated in his method by Thomson[76] in 1964 empirically. Later, Bradshaw[77] drew an analogy between bouyancy and "centrifugal" forces. The effects of Coriolis force on turbulence have been studied by Johnston[78]. Streamline curvature seems to influence principally the value of the momentum form factor and to a small extent the value of the momentum thickness. Thus, the TBL developing along a convex surface is brought closer to separation without altering appreciably its losses. An example of calculation of the effects of surface curvature is given in Figure 8. For the coriolis force effect, Bradshaw (Table I) cites the value $\frac{1}{80}$ for $\frac{\Omega\delta}{U_\infty}$ in order to produce a 10% change in skin friction coefficient or distance to separation. Johnston evaluates $\frac{\Omega\delta}{U_\infty}$ to be somewhat smaller than $\frac{1}{100}$ for turbines and of the order of $\frac{1}{10}$ for centrifugal compressors and pumps.

First order corrections can be made using Bradshaw's theory, taking into account that the constant of Monin-Oboukhou's meteorological formula used for surface curvature corrections takes the value 7 (convex surface), while the same constant measured for rotating flow[79] was found to have the value of 6.

(c) The effect of Free Stream Turbulence.

Bradshaw[40] points out that free stream turbulence in turbomachines consists of two parts, a part which is true turbulence and another which is introduced by unsteady effects (relative motion of the blades and their wakes). Free stream turbulence apparently doesn't influence laminar instability, but it influences transition and the subsequent development of the turbulent boundary layer[81]. Although measurements of turbulence in actual compressors are very difficult, the value of 10% is often advanced The effects of free stream "real" turbulence have been incorporated in the calculation method of Horlock and Lewkowicz[80] and the effects of time unsteady flow in the method of Bradshaw[82].

9. Conclusions.

The review that has been presented above reflects the present ignorance on turbulence and turbulent boundary layers along with the present inability to cope with the numerical problems posed, in spite of the positive steps that have been realized in BL research in the past decade and the advent of modern computing machines.

Only now the scientists allow themselves to be taken away from "academic" problems and concentrate on "real-life" ones. However, the lack of accurate experiments in almost all particular applications makes this task a difficult one.

REFERENCES

1. Liepmann and Roshko, Elements of Gasdynamics, J. Wiley and Sons (1957)
2. E. H. Kennard, Kinetic Theory of Gases, McGraw-Hill Co. (1938)
3. H. Schlichting, Boundary Layer Theory, Pergamon Press (1968)
4. C. C. Lin, Editor, High Speed Aerodynamics and Jet Propulsion - Turbulent Flows and Heat Transfer, Vol V, Princeton University Press (1960)
5. F. K. Moore, Editor, High Speed Aerodynamics and Jet Propulsion, Theory of Laminar Flows, Vol. IV, Princeton University Press (1964)
6. S. I. Pai, Viscous Flow Theory, Volumes I and II, D. Van Norstrand Co. (1957)
7. S. J. Kline, M. V. Morkovin, G. Sovran, D. J. Cockrell, Editors, Computation of Turbulent Boundary Layers - 1968, AFSOR-IFP-Stanford Conference, Proceedings, Vol. I, Thermoscience Div., Dept. of Mech. Eng., Stanford Univ., (1968).
8. M. V. Morkovin, Lectures on Transition, V.K.I. Lectures Series, Von Karman Institute Publications, 1967.
9. D. E. Coles, E. A. Hirst, Editors, Computation of Turbulent Boundary Layers - 1968, AFOSR-IFP-Stanford Conference, Proceedings, Vol. II, Thermoscience Div., Dept. of Mech. Eng., Stanford Univ. (1968).
10. N. Tetervin and C. C. Lin, A General integral form of the boundary layer equation for incompressible flow with an application to the calculation of the separation point of turbulent boundary layers, NACA Re 1046 (1951).
11. F. H. Clauser, Turbulent Boundary Layers in Adverse Pressure Gradients, Journal of the Aero. Sci. Feb. 1954 (p. 91)
12. J. C. Rotta, Turbulent Boundary Layers in Incompressible Flow, Progress in Aeronautical Sci., Vol.2 (1962).

13. P. Bradshaw, The turbulence Structure of Equilibrium Boundary Layers

14. K. Stewartson, Correlated Compressible and Incompressible Boundary Layers, Proc. Roy. Soc. A200, 84-100 (1949)

15. M. Morduchow, Analysis and calculation by integral methods of laminar compressible boundary layer with heat transfer and with and without pressure gradient, NACA Re 1245 (1955)

16. C. B. Cohen and E. Reshotko, The compressible laminar boundary layer with heat transfer and arbitrary pressure gradient. NACA Re 1294 (1956).

17. L. G. Loitsianskii, The Universal equations and Parametric Approximations in the Theory of Laminar Boundary Layers, P.M.M. 29, 74 (1965)

18. G. B. Thomson, A critical review of existing methods of calculating Turbule Boundary Layers, ARC 26,109, FM 3492 (1964)

19. A. A. Dorodnitsyn, General Method of integral relations and its application to boundary layer theory. Advances in Aero. Sci III, N.Y., Macmillan, 207-219.

20. H. E. Bethel, On the convergence and exactness of solutions of the Laminar Boundary Layer equations using an N-parameter integral formulation Galerkin-Kantorovich-Dorodnitsyn., ARL 60-0090, July 1966.

21. K. O. Felsch, Contribution to the calculation of turbulent boundary layers in two-dimensional incompressible flow. Dissertation T. H. Karsruhe and DLR-FB66-46.

22. P. Goldberg, Upstream history and apparent stresses in turbulent boundary layers. Gas Turbine Laboratory Report No. 85, MIT

23. H. McDonald and J.A.P. Stoddart, On the development of the incompressible turbulent boundary layer. Aero. Res. Coun. R and M, 3484.

24. J. F. Nash and A. G. J. Macdonald, A calculation method for the incompressible turbulent boundary layer, including the effect of upstream history on the turbulent shear stresses, NPL Aero Report 1234.

25. P. Bradshaw, D. H. Ferris, and N. P. Atwell, Calculation of boundary layer development using the turbulent energy equation, J. Fluid Mech. 28, 3, 593-616 (1967)

26. G. L. Mellor, Incompressible turbulent boundary layers with arbitrary pressure gradients and divergent or convergent cross flows, AIAA Jour. 5. 1510-1579.

27. S. V. Patankar and D. B. Spalding, A Finite difference procedure for solving the two-dimensional boundary layer equations, Int. J. Heat Mass Transfer, 10, 1389-1412

28. L. Rosenhead, Editor, Laminar boundary layers, Oxford 1963.

29. L. G. Loitsianskii, The universal equations and parametric approximations in the theory of the laminar boundary layer P.M.M. 29, 74 (1965)

30. P. Bradshaw, Boundary-Layer Problems of 1966 NPL, Aero Report 1203, A.R.C 28191 (1966)

31. A.M.O. Smith and D. W. Clutter, Machine calculation of compressible laminar boundary layers, AIAA J. 3, 639 (1965)

32. J. Ginoux, Editor, Short Course on Laminar and Turbulent Flows, Lecture Series, VKI Publication (1968)

33. H. J. Obrenski and others, A portfolio of Stability Characteristics of incompressible boundary Layers, AGARDograph 134 (1969)

34. E. R. Van Driest, Recent Studies in Boundary Layer Transition, Int. J. Eng Sc. Vol 13, pp. 341-358.

35. K. D. Papailiou, Optimization d'autage de compresseurs a' fort charge sur la base des theoreis des couches limites. VKI TN 61 (1969)

36. Mozol'kov, A method for determinating the transition from a laminar to a turbulent boundary layer, NASA AD 696307 (1969)

37. H. R. Head, Improved entrainment method for calculating turbulent boundary layers, ARC 31043 (1970)

38. Mikolajczak, Private communication (tests performed at Pratt and Whitney Co

39. J. Le Foll, A theory of representation of the properties of boundary layers on a plane, Proc. of Seminar on Advanced Problems in Turbomachinery, VKI, March 23-30, (1965)

40. P. Bradshaw, Prediction of turbulent shear layers in Turbomachines. International Symposium on the Fluid Mechanics and Design of Turbomachinery, Pennsylvania State University, September 1970.

41. P. Bradshaw, Special topics in Turbulent Flows, VKI short course on Turbulent Boundary Layers, VKI Publications (1969)

42. Powers, A review of factors affecting boundary layer transition, NASA TN D3384 (1966)

43. Chan, An approximate method for three-dimensional compressible laminar boundary layers with small cross-flow, N.R.C. LR 455 (1966)

44. Smith, Approximate Solutions of the 3D laminar boundary layer momentum integral equations ARC CP 1064 (1969)

45. J. C. Cooke and M. G. Hall, Boundary Layers in three dimensions, Proc. Aero. Sci. 2, 221 (1962)

46. G. S. Raetz, A method of calculating three-dimensional laminar boundary layers of steady compressible flows, Northrop Aircraft Inc., Refp. No. NAI-58-73 (1957)

47. J. Der and G. S. Raetz, Solution of general three-dimensional laminar boundary layer problems by an exact numerical method, IAS Paper No 62-70 (1962)

48. Krause, Finite difference equations for calculating 3D boundary layers, DLR Re 69-66 (1969)

49. Smith, Calculation methods for three-dimensional turbulent boundary layers, ARC 28586 (1966)

50. Cumsty and Head, The calculation of 3-dimensional turbulent boundary layers, ARC 28180 (1966), CP 1077 (1970)

51. G. E. Gadd and W. H. H. Banks, A preliminary report on boundary layers on screw propellers and simpler rotating bodies, ARC 23423-F.M. 3150 (1962)

52. H. R. Head, Short Course on Turbulent Boundary Layers, Part 3. VKI Publications (1968)

53. P. Bradshaw, Calculation of boundary layer development using the turbulent energy equation. VII. Three-dimensional flow, NPL Aero Re 1286 ARC 30910 (1969)

54. Pal, Viscous Flow along a corner. I Asymptotic features of the corner layer equation PIB 69-18 (1969)

55. Rubin, Viscous flow along a corner. II Numerical solution of the corner layer equations, PIB Re 69-33

56. H. J. Perkins, The corner boundary layer, Eng. Dep., Cambridge Un. UCT/12 (1968)

57. K. Stewardson, On Goldstein's theory of laminar separation, Quart. J. Mech Appl. Math. 11, 399-410 (1958)

58. B. S. Stratford, Flow in the Laminar Boundary Layer Near Separation

59. S. J. Kline, Some new Conceptions of the Mechanism of Stall in the Turbulent Boundary Layer, Jour. Aero. Sci. Vol. 24, pp 470-471 (1957)

60. S. J. Kline, On the nature of Stall, Jour. of Basic Eng. Trans. ASME, Ser. D. Vol. 81, p 305 (1959)

61. J. E. Green, Interaction Between Shock Waves and Turbulent Boundary Layers Lecture Series 10, VKI Short Course on Turbulent Boundary Layers, VKI Publications (1969)

62. C. Y. Liu, Boundary Layer Separation, Ph.D. Dissertation, Colorado State U versity, 1967.

63. J. J. Giroux, Editor, Separated flows, a round table discussion, AGARD R 5 (1966)

64. Howarth, Analytical investigation of laminar separations using the "Crosco- mining parameters" method. NASA CR 745 (1966)

65. Gregory, Progress report on observations of three-dimensional flow pattern obtained during stall development on airfoils and on the problem of measuring two-dimensional characteristics. NPL Aero Re 1309 (1970)

66. V. A. Sanborn, Boundary Layer Separation and Reattachment. International Symposium on the Fluid Mechanics and Design of Turbomachinery. Pennsylvan State University (1970)

67. A. Buri, Eine Berechnungsgrundlage fur die turbuleute Grenzschicht bei beschleunigter und rerqogerter Grundstromung. Diss. 1931 Eidgen Techn. Hochschule Zurich.

68. E. C. Maskell, Flow Separation in Three Dimensions, Rep. Aero 2565, British RAE (1955).

69. B. S. Stratford, The Prediction of Separation of the Turbulent Boundary Layer, J. Fluid Mech. 5, 1-16 (1959)

70. J. Lieblein, C. Francis and L. R. Broderick, Diffusion Factor for Estimating losses and limiting blade loading in Axial Flow Compressor Blade Elements, NACA RM E53D01 (1953)

71. S. Lieblein, Loss and Stall Analysis of Compressor Cascades, ASME Trans. Journal of Basic Eng. Sept 1959, p. 387.

72. P. Bradshaw, Short Course on Turbulent Boundary Layers, Part 2, VKI Publication (1968)

73. D. Coles, The turbulent boundary layer in a compressible fluid , Rand Rep. R-403-PR (1962)

74. Patel and M. R. Head, Some observations on skin friction and velocity profiles in fully developed pipe and channel flows, J. Fluid Mech. 38, p. 181 (1969)

75. B. S. Masey, B. R. Clayton, Laminar boundary layers and their separation from curved surfaces, ASME J.B.E., p 483 (discussion) (1965)

76. B. G. Thomson, The new entrainment method, AGARDOgraph 97 (1965)

77. P. Bradshaw, The analogy between streamline curvature and buoyancy in turbulent shear flow, NPL Aero Re. 1231 (1967)

78. J. P. Jonston, The effects of Rotation on Boundary Layers in Turbomachine Rotors.

79. P. Bradshaw, Private Communication (1970)

80. A. K. Lewkowicz and J. H. Horlock, Report ULME/B.19, Dept. of Mechanical Eng. Liverpool Un. (1966).

81. S. J. Kline, A. V. Lisin and B. A. Waitman, Preliminary Experimental Investigation of Effect of Free-Stream Turbulence on Turbulent Boundary Layer Growth, NASA TND-368 (1960)

82. P. Bradshaw, Calculation of Boundary Layer Development using the Turbulent Energy Equation IV: Unsteady Flow, NPL Aero. Rep. 1288, ARC 30912 (1969).

Table I

Strength of "special effects" needed to change surface shear stress or distance to separation by ten percent.

Special Effect	Order of Magnitude
Sweepback (with given chordwise pressure gradient).	= 45 deg.
Longitudinal curvature (camber)	$\delta/R = 1/80$ or 35 deg. turning angle
Rotation (component about spanwise axis).	$\frac{\Omega\delta}{U_\infty} = 1/80$
Low Reynolds number. (c_f compared with Schoenherr value).	$\frac{U_\infty\theta}{\nu} = 650$ ($U_\infty x/\nu = 3\times10^6$)
Free stream turbulence (small scale)	3%

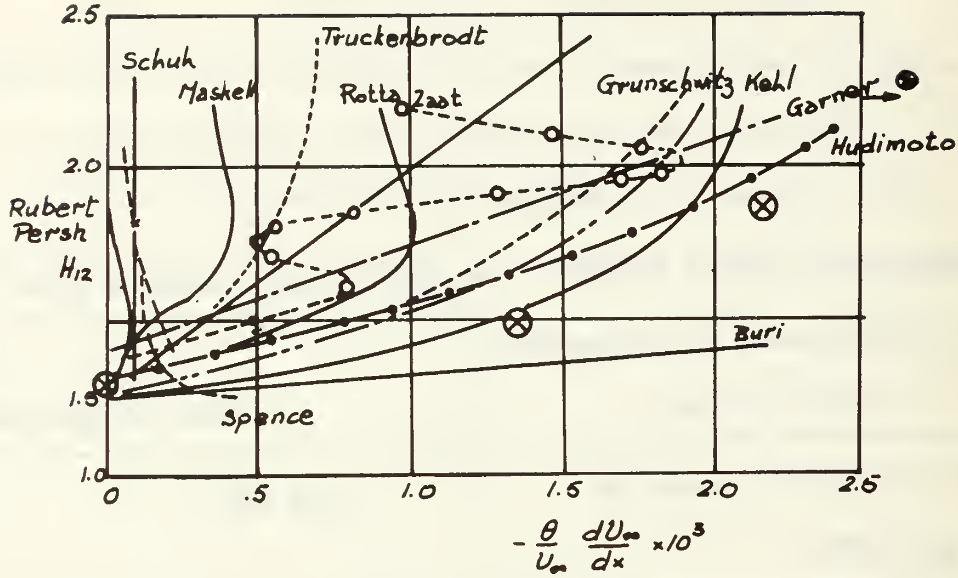


Figure 1. Comparison of experimental results from equilibrium boundary layers and theoretical prediction methods (ref. 12) $\frac{U_\infty \theta}{\nu} = 10^4$

⊗ Flat plate boundary layer and equilibrium layers from Clauser's measurments

⊕ Equilibrium layer for zero wall stress of Stratford.

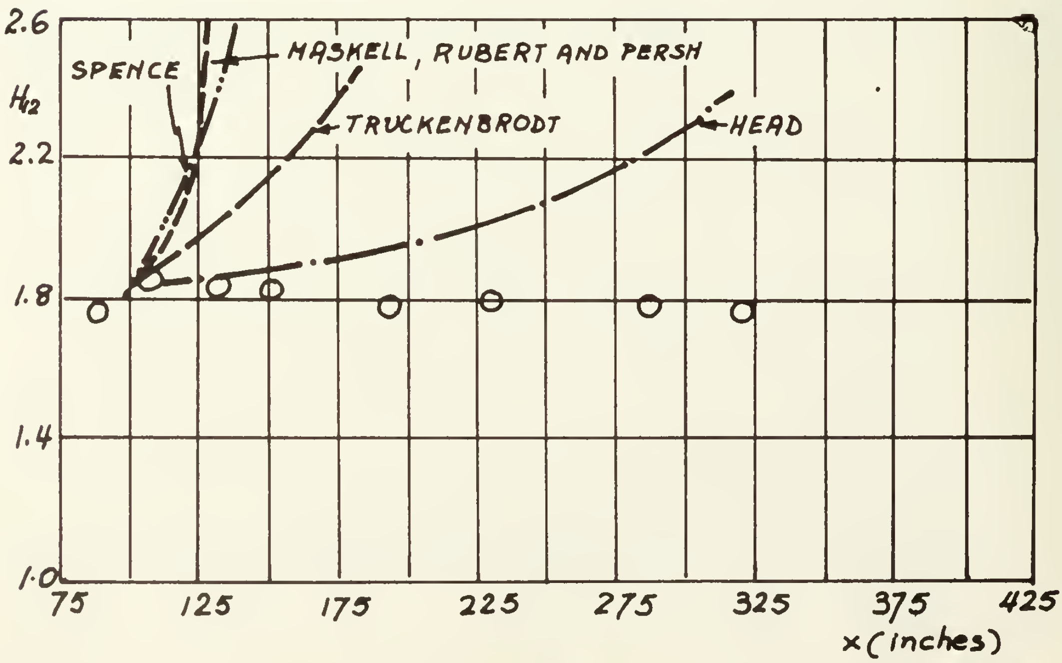


Figure 2. Comparison of experimental results and Theoretical prediction methods. Clauser, series II (ref. 18)

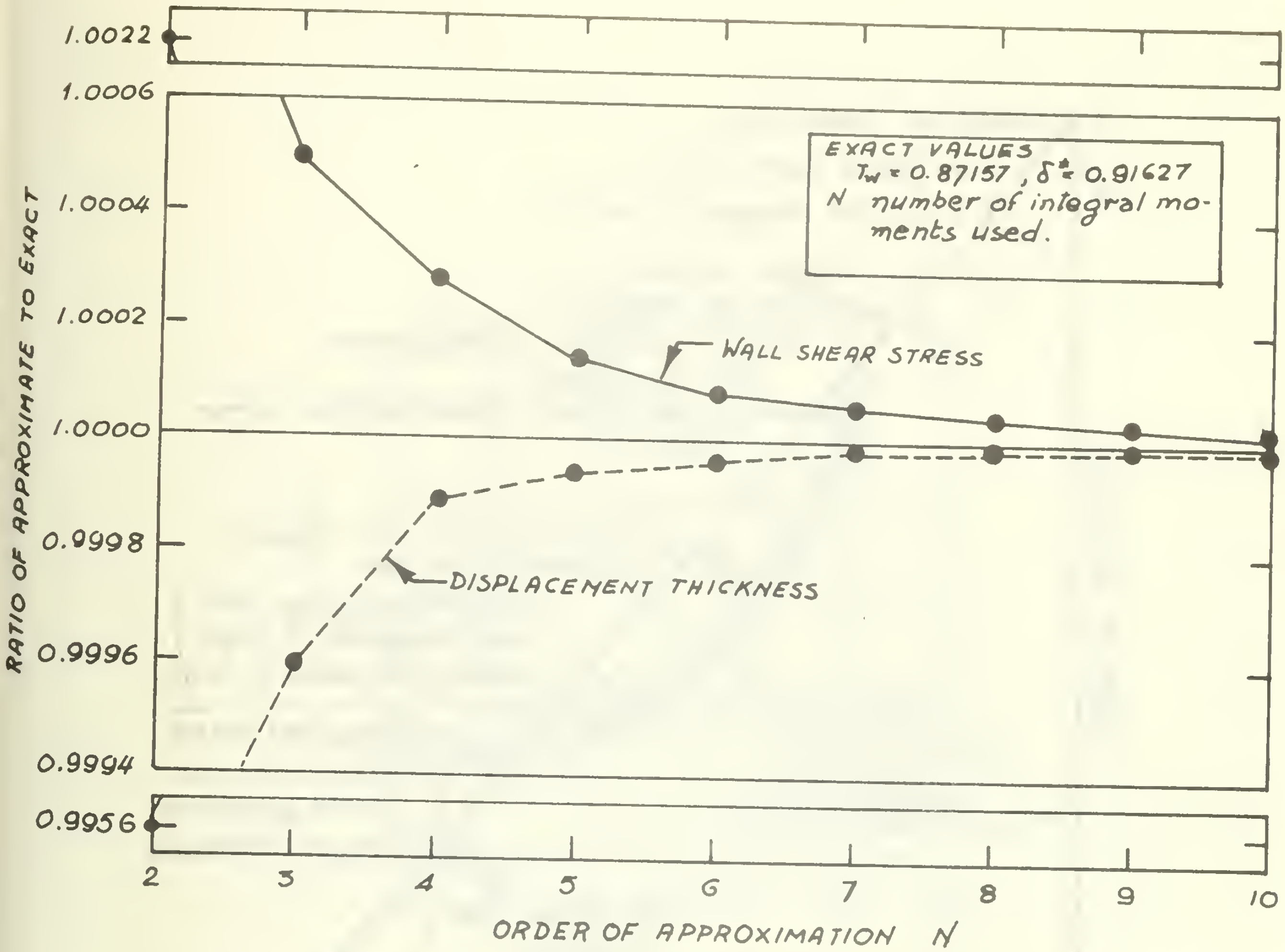


Figure 3. Comparison of the Galerkin-Kantorovich-Dorodnitsyn approximate solution to the exact one. Stagnation Point Flow. (ref. 20)

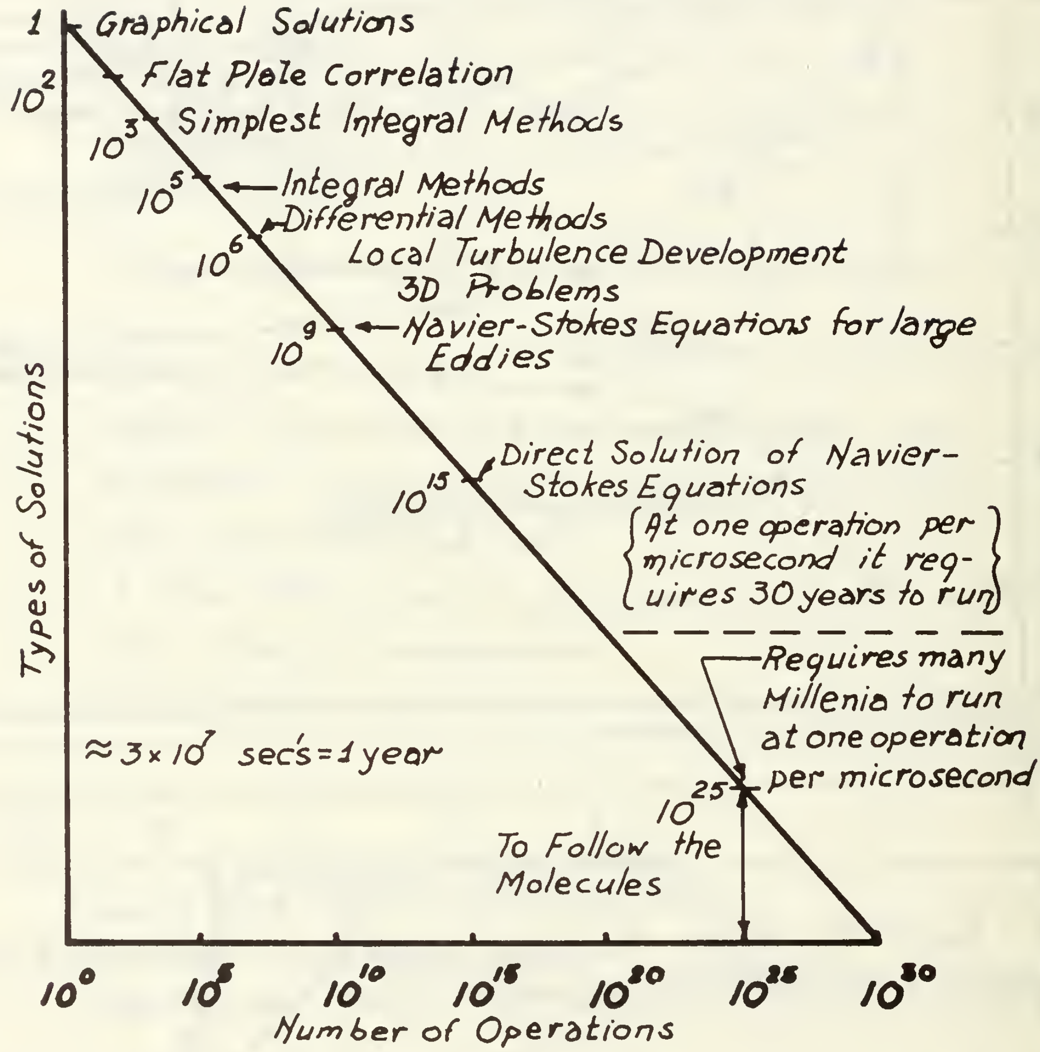


Figure 4. Number of operations required for the numerical solution of various problems (ref. 7).

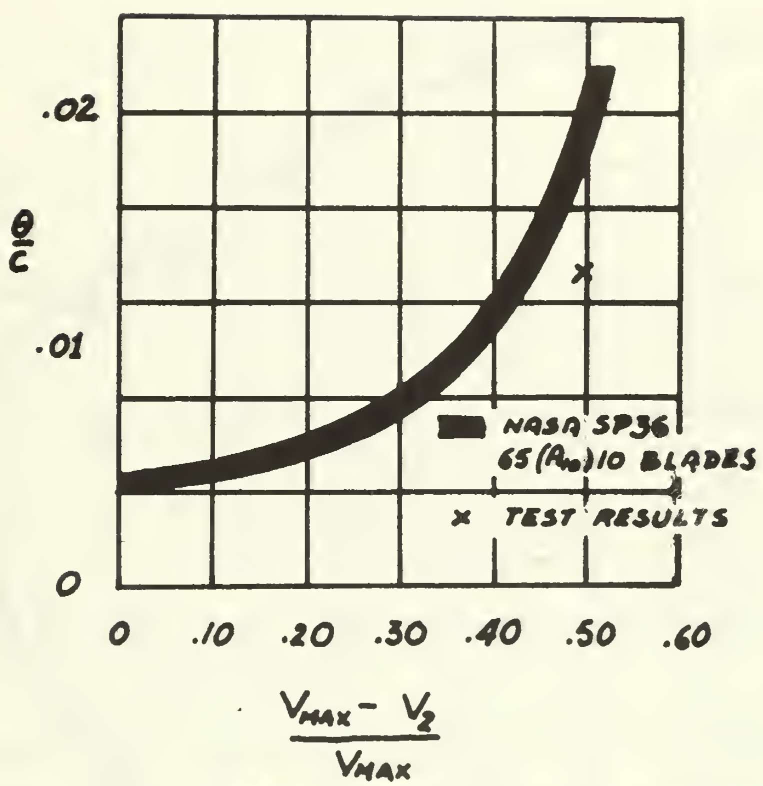


Figure 5. Test results for the blade designed by the method of ref. 35

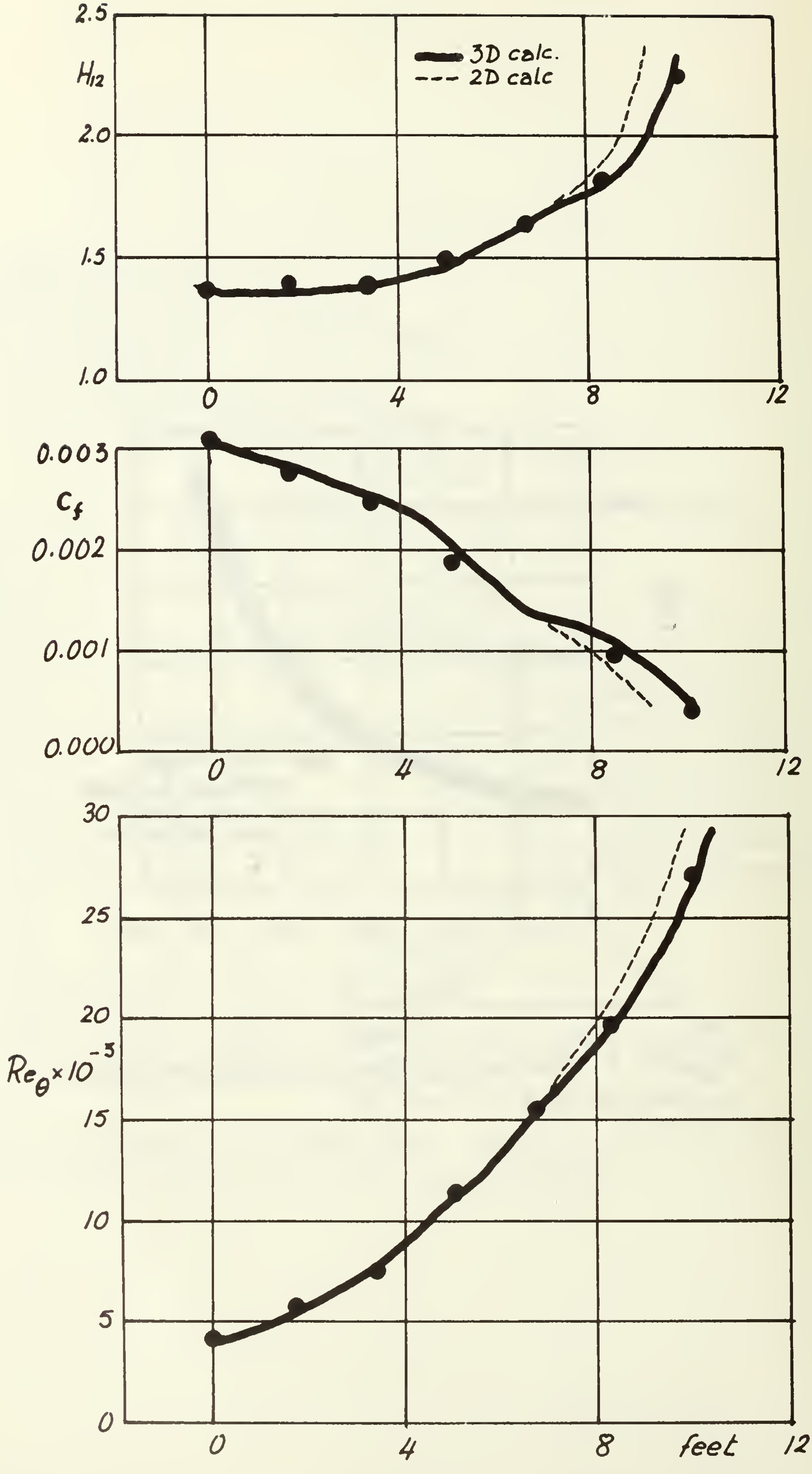


Figure 6. Comparison of 2D prediction methods with experimental results (Head's method, ref. 37).

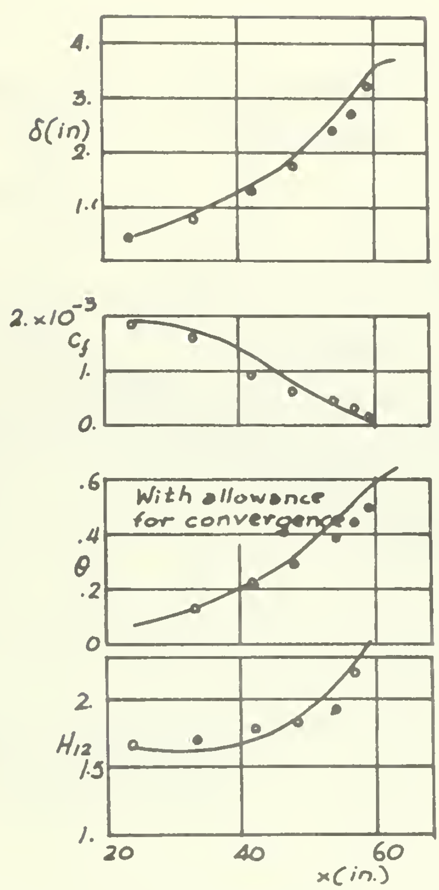


Figure 7. Comparison of 2D prediction methods with experimental results. (Bradshaw's method, ref. 37).

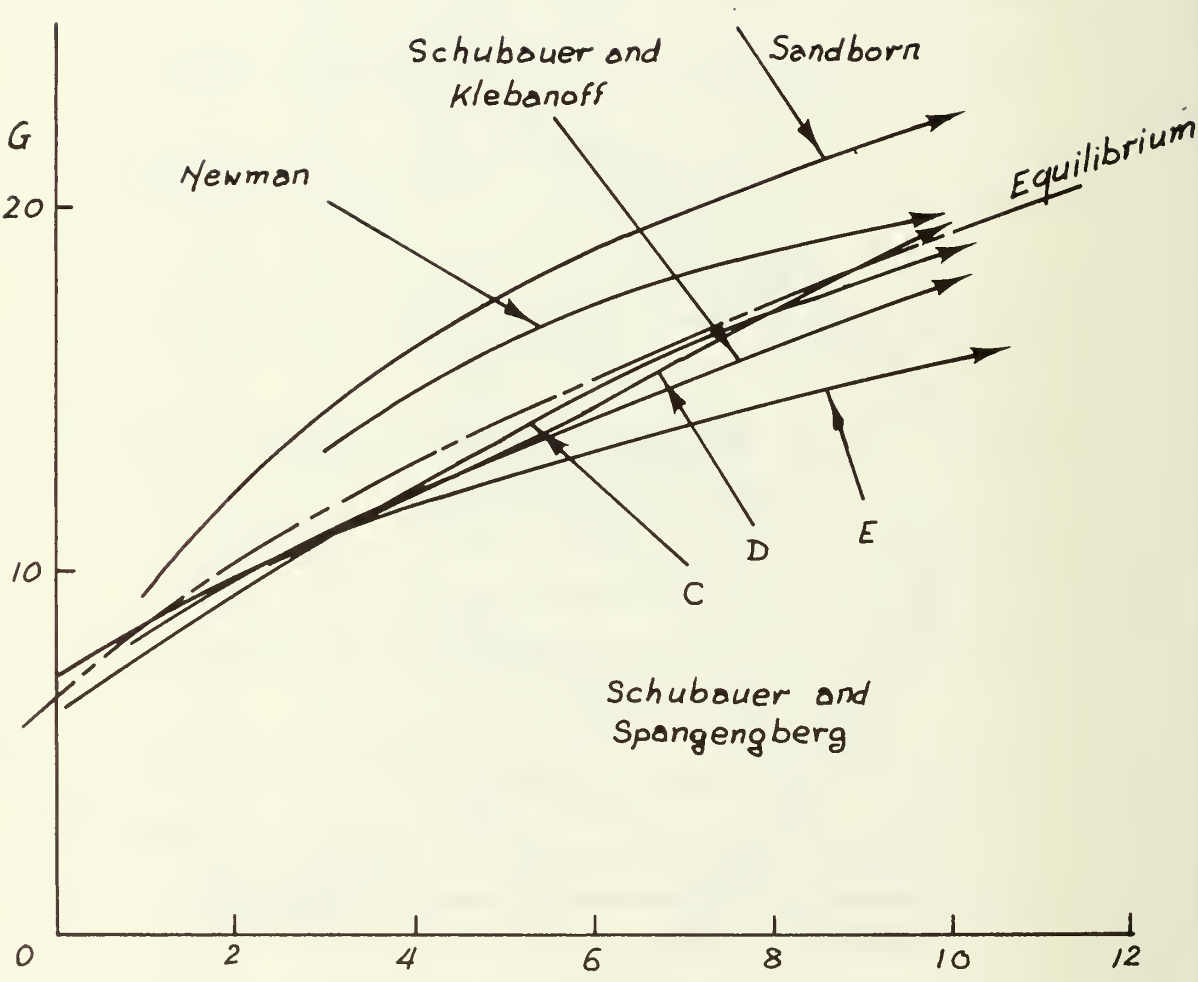


Figure 8. Turbulent boundary layer trajectories for increasing pressure gradient.

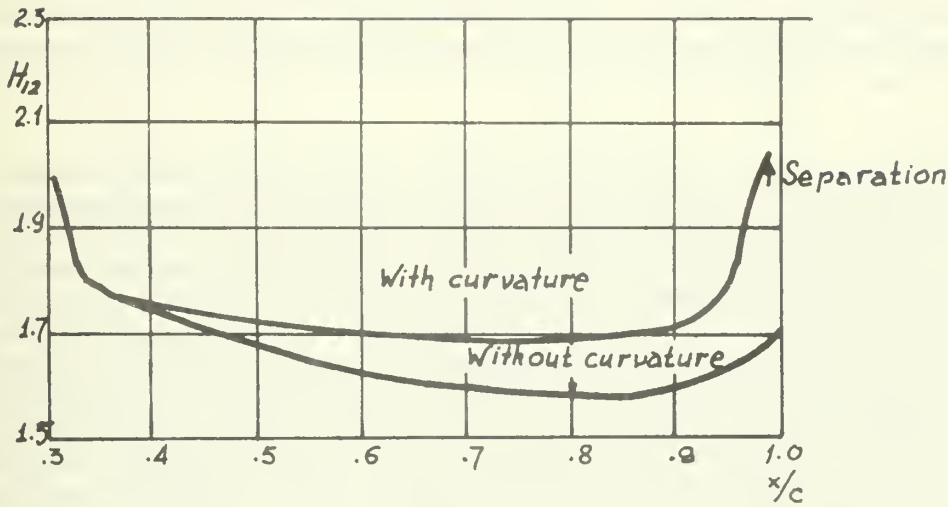


Figure 9. Effects of surface curvature. Momentum form factor of the turbulent part of the boundary layer developing along the suction surface of a 45° turning compressor blade.

DISCUSSION

(Abbott) One way to solve partial differential equations is to employ a finite difference scheme, another way is to employ an integral technique. By both techniques, what finally results is a system of algebraic equations. There has been work (R. Vichnevetsky: "Functional Approximation Methods Applied to the Simulation of Field Problems," Approximation Methods for Solving Engineering Problems, R. H. Kohr and D. E. Abbott, Purdue University Short Course Lectures, July 1968. And N. B. Ferguson and B. A. Finlayson: "Development of Orthogonal Collocation for Nonlinear Partial Differential Equations," Dept. of Chem. Engrg., University of Washington, Seattle, Wash., 1969.) that shows that in this respect the integral and differential methods are completely equivalent. You referred also to some of the conclusions from the Stanford Conference, for example, that much of the data had three-dimensional effects.

The criterion that was used at Stanford to establish when a given flow is two-dimensional or not was the integral equation for two-dimensional flow, that is, neglecting curvature and many other things. You've already mentioned that curvature corrections are important and others, such as myself and Cebeci and Smith, have shown that for many of the flows which are ruled to be at least pretty good border line cases as regards three dimensional effects, when curvature is included, the two-dimensional calculations do pretty well. Thus some of the questionable Stanford data would have to be very carefully reconsidered before throwing it out. I think also that you are being a little bit over pessimistic, particularly for this meeting, on really what the state of the boundary-layer calculation art is because there are a number of very good production-type methods that you can use off the shelf, which include curvature, low Reynolds number effects, transpiration etc., which are very accurate. These are production-type methods which include a criterion for the calculation of transition and also some criteria for separation. I am refering to the Mellor and Herring programs and the Cebeci and Smith programs (which also include transverse curvature). They are not answering questions about detailed phenomenological assumptions, but they can make successful predictions with engineering accuracy for integrated boundary-layer parameters.

(Papailiou) Thank you for your comments. I can only remark that the instability point can be calculated up to a Mach number of 2.5 more or less with accuracy, although there are still uncertainities for supersonic Mach numbers. I am not, however, ready to admit that there is any kind of transit criteria which will calculate transition and which can be applied by anybody without the "know-how" of the one who invented them for either incompressibl or compressible flow. Also, I feel that if you have a two-dimensional boundary-layer calculation method and you cannot from there extend the empirical relations that you have in order to calculate a three-dimensional turbulent boundary layer, the problem remains essentially unsolved.

(Huffman) I think that we are misinterpreting the outcome of the Stanford conference. The correct interpretation of the result of that evaluation is that there are a number of methods that will adequately predict the available data which are judged acceptable, and it is not correct for one to interpret the success or failure of a physical hypothesis on the basis of good results because if anything is illustrated by those results, it is that you cannot make that interpretation. In general, we can conclude that it is very easy to predict the mean velocity and very difficult to predict the turbulent shear stress. And if you look at it in more detail you can find that you can do a pretty poor job with the turbulent shear stress and still do a very good job with velocity. So what this means is that in general the shear stress is a much more sensitive parameter to predict than the velocity. We should, if we are aiming at unearthing the physical hypothesis, design experiments that test a specific hypothesis. However, purely two-dimensional incompressible boundary layers don't appear to do this. I am talking about the detailed shear stress profile. I am talking about the basic hypothesis that underlies all these empiricisms. The correct interpretation of these equations is mainly a calculating framework. The only advantage that the new methods have that the old methods didn't have is that they maximize the information content available while they minimize the empirical influence. You really can't make any comment any broader than that about the physics. If you talk to Peter Bradshaw, he won't make any interpretation much broader than that.

(Abbott) There is another game you can play on much of the Stanford data. If you consider all of the cases, and proceed to make calculations as one would for laminar flow by assuming only a turbulent velocity profile and use an integral method (that is, without employing a wall shear-stress model), it is possible to satisfactorily calculate much of the experimental data and to do just as well as many of the presented methods with all their different physics. Not all of them, but many of them; (V. G. Forsnes and D. E. Abbott: "A Unified Comparison of Local and Global Turbulent Shear Stress Models Utilized in the Prediction of Two-Dimensional, Incompressible Turbulent Boundary Layers," School of Mechanical Engrg., Technical Rept. FMTR-69-4, Purdue University, 1969. And G. R. Deboy and D. E. Abbott: "Examination of Turbulent Shear Models and the Prediction of Compressible Turbulent Boundary Layers by the Method of Weighted Residuals," School of Mechanical Engrg., Technical Report FMTR-71-1, Purdue University, 1971.)

(Huffman) But you can deduce experiments for which that is not true, and people are doing that at the present time.

(Abbott) In many of the cases we are not critical judges of physics or methods.

(Huffman) But as far as the bulk of the attendees are concerned, that is a very positive result. It is extremely positive. That tells you that, in general, these methods will do a good job for you in certain cases without really having much physics in them.

(Papailiou) We can put the above into the following statement. A basic contribution of the Stanford Conference is that it has established that we can calculate with confidence a two-dimensional turbulent boundary layer, but still that this calculation cannot contribute to the physics in order to be able to go and calculate the three-dimensional one.

(Huffman) That is correct. That is the correct interpretation to imply.

(Fagan) We have used the Mellor-Herring Programs starting out with laminar boundary layers and empirical transition rules of going through to turbulent boundary layers and calculated the losses for a blade cascade, and we have accurately reproduced what we measured in a wind tunnel. We spend a lot of money on the two-dimensional cascade studies, so I think that the two-dimensional boundary layer problem as an application to this case was good, as it can save us a lot of money.

(Papailiou) In some calculations I have done, using an empirical transition criterion, I reproduced also experimental results. But that doesn't prove anything.

(Fagan) I don't know how well we reproduced the transition, but we had the right losses for that blade cascade.

(Olson) I would like to inject a little bit of optimism here about being able to calculate transitional boundary layers with the same kind of vigor as we are now talking about for the turbulent case using an extension of the turbulence kinetic energy equation. We have been able to predict heat transfer through transition which is a better test of the theory than the losses. I think we now can begin to be optimistic about the possibility of computing boundary layers with numerical techniques with rather sophisticated turbulence models based on turbulence kinetic energy equations from a stagnation point through the transition of the flow into turbulence. I think some of this work will be published in the near future. There is a lot of work going on in this area; and I think that from the results, there is reason to be very optimistic.

(Mikolajczac) I might make one comment from an engineer's standpoint. The calculation methods we have for two-dimensional minimum losses are more than adequate. The problem to be solved, however, is to be able to predict loss and turning over the full incidence range. To predict the performance in stall and choke, we need a valid separation criterion; and until this is established, the boundary layer analysis will be of limited use to compressor design.

(Papailiou) I would like also to ask the people that do the transition calculations how they calculate the initial conditions for the turbulent boundary layer.

(Huffman) No one has been able to do that successfully.

(Herring) Well, not using the turbulent energy equation.

(Lakshminarayana) How does one use this boundary layer information to correct the potential flow field?

(Papailiou) This problem has been more or less solved, but only when separation is not present. Temple and Preston did the original work, but for a single airfoil. Since then, there has been some work done at NPL and very recently at Betford (RAE). However, all this work concerns flows where separation is absent.

(Huffman) I would like to make a general comment on separation. I think that when you made the comment that boundary layer equations aren't valid, what is meant by that is that the underlying assumptions that are used to reduce the elliptic equations into the parabolic equations are no longer valid. Now, whether or not the solutions correspond to experimental results remains to be seen.

(Papailiou) I wouldn't say that. I would say simply that the assumptions of Prandtl are not valid, as you don't know if you get parabolic or hyperbolic or even elliptic equations because in certain formulations for three-dimensional flows you get elliptic equations.

(Huffman) Let me make one more comment. Spalding at Imperial is directing the bulk of his energies now on solving the elliptic equations. He is looking primarily at separation. He is solving these equations numerically using an approach that is similar to the technique that he used for the parabolic equations.

(Olson) We have reasoned along the same lines as Spalding to solve the Navier-Stokes equations for laminar flow in the region of the separation point. And we have shown that there is a difference; the separation point predicted by the boundary layer theory is different from that which you predict with the Navier-Stokes equations. There seems to be a significant elliptical upstream influence.

(Huffman) That is true. This is the result Spalding had. He has been able to predict separated flow over a step, and there is a very strong elliptic influence. In fact, there is a secondary boundary layer formed on the bottom of the step.

NEW MEASURING AND
FLOW-VISUALIZATION TECHNIQUES

by

Dr. D. J. Collins
Professor
Naval Postgraduate School

Reprinted from:

6-1

JOURNAL OF APPLIED PHYSICS VOLUME 42, NUMBER 3 1 MARCH 1971

Determination of Three-Dimensional Density Fields from Holographic Interferograms*

ROBERT D. MATULKA† AND DANIEL J. COLLINS
Naval Postgraduate School, Monterey, California 93940

(Received 16 July 1970)

The successful application of holographic interferometry, and an associated mathematical reduction process, to the determination of an asymmetric three-dimensional density field of an aerodynamic phenomenon is reported. An integral inversion method from the field of plasma physics has been extensively evaluated by applying it to the determination of functions, both axisymmetric and asymmetric, which simulate aerodynamic density fields. The application of holographic interferometry has been extended to provide multiple holograms about a test region, with sufficient coverage to provide interferometric data for the successful solution of the density field. The analytical and experimental methods developed were applied to an experimental axisymmetric test field, the supersonic flow from a free jet, and shown to be comparable to a previous solution obtained by the Abel inversion method. Further, the free jet was tilted to provide a test field which was asymmetric in the plane of solution. Comparison of the resulting asymmetric solution was shown to be consistent with the previously obtained axisymmetric solution.

I. INTRODUCTION

Holography has enabled the aerodynamicist to "freeze" the interferometric view of a transient phenomenon and subsequently to view the field in three dimensions, as through a "window." This new capability has indicated the possibility of quantitatively determining the density in such a field, with no symmetric restrictions on its form.

To make quantitative determinations of asymmetric density fields, it was necessary to

(1) Invert the fringe number functions that describe the interferometric data,

$$g(y', \xi) = \int_{x_1'}^{x_2'} f(x, y) dx' \qquad (1)$$

to obtain the asymmetric density function f from within the integral. In the search for such an inversion process the possibility of using an asymmetric line integral inversion scheme which had been developed in the field of plasma emissivity (Maldonado, 1966),[1] became evident. The method was evaluated by extensive computerized testing on density functions found in aerodynamics, including shock waves. The inversion method was found to be quite accurate, giving results which were generally within 1% of the test function.

(2) Obtain holographic interferograms of sufficient angular coverage about a test region to provide the interferometric data required by the inversion process. The method requires 180° viewing angle for fully asymmetric fields. In the present experiments, 90° was sufficient because of a plane of symmetry in the flow. To obtain the necessary experimental data, a holographic work table was constructed and used to design the necessary optical arrangement. The capability of achieving multiple Q-switched holographic interferograms about a test region, with no opaque objects present, was demonstrated.

(3) Apply real holographic interferometric data to an inversion process in order to demonstrate the practicality of the technique. A stepwise evaluation of the system included an axisymmetric evaluation of a free jet for comparison with the previously available solution of Winckler (1948).[2] Having attained a solution for the three-dimensional axisymmetric field, the asymmetric capability of the technique was tested by tilting the free jet, which destroyed the axisymmetry in the plane of inversion. The resulting data were inverted asymmetrically and compared with the axisymmetrically measured density. The result is shown to be a self-consistent comparison of the asymmetric inversion.

A. Holographic Interferometry

The application of holography to interferometry (Heflinger *et al.*, 1966)[3] derives from the fact that the holographic film records the diffraction pattern almost linearly. When double exposed; the hologram records two diffraction patterns superimposed upon one another. Therefore, when a double-exposed hologram is reilluminated, each recorded diffraction pattern will diffract its own first order beams. An observer will see both recorded scenes simultaneously. To produce a holographic interferogram of a test section, one exposes the hologram for one-half the exposure time with no flow in the test section (the uniform field provides the comparison beam); then completes the exposure with the subject flow field present. The two reconstructed waves will interfere with each other in much the same manner as do the two waves of the Mach–Zehnder in the infinite fringe configuration.

The chief advantage of this method is that, except for differences in the test section, both the test and the comparison beams have traveled through exactly the same optical regions. Optical components are then automatically matched. In practice rather crude optical components can be used with excellent results.

By a slight rotation of the hologram or of one of the mirrors between exposures, one can achieve the same finite fringe presentation as with the Mach–Zehnder.

B. Light Field Interferograms

True three-dimensional interferograms are obtained when a diffuser plate is placed between the source and

Editor's comment: This is the paper on which Professor Collins based his presentation.

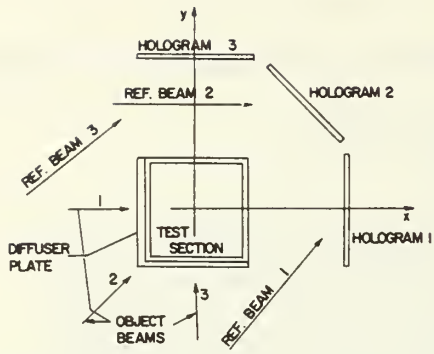


FIG. 1. Hologram arrangement for wide angular range of views.

the test section. The object beam then becomes diffused through the test section and appears to the observer as a continuous background of source points against which the test section, and its interference pattern, become a silouette. Because each point of the diffuse plate acts as an individual source for any line of sight passing through it, Eq. (3) may be evaluated for any line which passes through both the diffuser plate and the hologram. A continuous evaluation of the function g can thus be provided as a function of position and angle. This function g, the fringe pattern, changes as the observer changes viewing aspect, but generally the fringe patterns cannot be localized by their parallax.

By arranging several holographic plates about a test section, as in Fig. 1, one can obtain the fringe number function for a rather continuous segment of angular variation about the z axis of the field. The resulting array of integral values can be applied to the inversion of the fringe integral equation to provide a solution for the density field in the test section.

II. THEORETICAL APPROACH

A. The Integral Inversion

The basic equation to invert is Eq. (1). Rewritten for a plane of constant z, one has

$$g(y', \xi, z_c)=Q\int_{x_1'}^{x_2'} f(x, y, z_c)dx', \tag{2}$$

where

$$f(x, y, z_c)=[\rho(x, y, z_c)/\rho_\infty]-1 \tag{3}$$

and

$$Q=\rho_\infty\beta/\rho_s\lambda. \tag{4}$$

x' and y' are measured in a coordinate system which is rotated by an angle ξ about the z axis (Fig. 2).

The index of refraction is a function of density given by

$$n=1+\beta\rho/\rho_s$$

thus β is equivalent to a dimensionless Gladstone–Dale constant and ρ_s, the reference density, is taken at standard conditions.[4]

The integral inversion method utilized in this investigation was first reported by Maldonado *et al.* in 1965[5] (also see Ref. 6). It was used for obtaining plasma emissivity within a particular region from the measured values of emission intensity measured from outside the region. The form of the equation resulting from such emissivity studies is identical to that of Eq. (2). The procedure involves the representation of the function f of Eq. (2) in a complete set of orthogonal functions, with the expansion coefficients evaluated by use of the orthogonality condition.

The function f is assumed to be squared integrable over the infinite plane so that it may be expanded in a complete set of orthogonal functions.

The unknown function may be expanded in a special set of functions $U_{m+2k}^{\pm m}$ which are orthogonal with respect to the weighting function $\exp[-(\alpha^2x^2+\alpha^2y^2)]$

$$f(x, y)=\sum_{m=0}^{\infty}\sum_{k=0}^{\infty}\epsilon_m\{C_{m+2k}^{m}(\alpha)U_{m+2k}^{m}(\alpha x, \alpha y)$$
$$+C_{m+2k}^{-m}(\alpha)U_{m+2k}^{-m}(\alpha x, \alpha y)\}\exp[-(\alpha^2x^2+\alpha^2y^2)], \tag{5}$$

where $\epsilon_m=\frac{1}{2}$ for $m=0$, $\epsilon_m=1$ for $m=1, 2, 3, \cdots$, and $C_{m+2k}^{\pm m}$ are the unknown coefficients of the expansion. α is an arbitrary scale factor which may be considered the reciprocal of a non-dimensionalizing coefficient.

The functions $U_{m+2k}^{\pm m}$ are defined

$$U_{m+2k}^{\pm m}(\alpha x, \alpha y)=(-1)^k\alpha[k!(\alpha^2x^2+\alpha^2y^2)^m/\pi(m+k)!]^{1/2}$$
$$\times\exp(\pm im\phi)L_k^m(\alpha^2x^2+\alpha^2y^2), \tag{6}$$

where $\phi=\tan^{-1}(y/x)-(\pi/2)$ and L_k^m is the associated Laguerre polynomial

$$L_k^m(\alpha^2x^2+\alpha^2y^2)=\sum_{s=0}^{k}\frac{(m+k)!}{(k-s)!(m+s)!s!}$$
$$\times[(-1)(\alpha^2x^2+\alpha^2y^2)]^s. \tag{7}$$

The function $U_{m+2k}^{\pm m}$ has a gauss transform:

$$I_{m+2k}^{\pm m}(\alpha y', \xi)=\int_{-\infty}^{\infty}U_{m+2k}^{\pm m}(\alpha x, \alpha y)\exp(-\alpha^2x'^2)dx'$$
$$=\frac{\exp(\pm im\xi)H_{m+2k}(\alpha y')}{[k!(m+k)!]^{1/2}2^{m+2k}}, \tag{8}$$

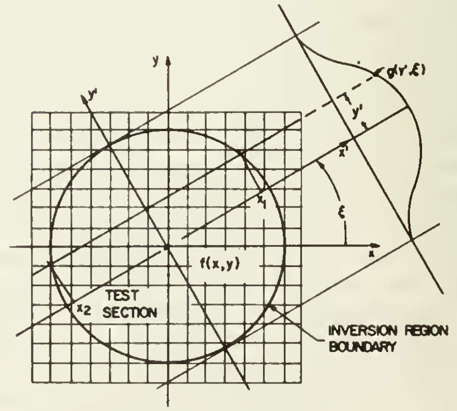


FIG. 2. Basic coordinate system of inversion process.

where $H_{m+2k}(\alpha y')$ are Hermite polynomials of order $m+2k$. The particular advantage of the set of functions $U_{m+2k}^{\pm m}$ is that they are "invariant in form" to a rotation of coordinate system (Maldonado).[7] That is, they remain an orthogonal set under a rotation of the coordinate system. Observe that in the equation for the polynomial (6), the angle ϕ occurs only in the complex exponential term.

In terms of the expanded function f, and utilizing the transform relation of Eq. (8), Eq. (2) becomes

$$g(y',\xi)=\sum_{m=0}^{\infty}\sum_{k=0}^{\infty}\epsilon_m[k!(m+k)!2^{2m+2k}]^{-1/2}$$
$$\times[C_{m+2k}^{m}(\alpha)\exp(im\xi)+C_{m+2k}^{-m}(\alpha)\exp(-im\xi)]$$
$$\times H_{m+2k}(\alpha)\exp(-\alpha^2y'^2). \quad (9)$$

Equation (9) is subject to the following orthogonality condition:

$$\int_{-\pi}^{\pi}\exp(\pm im\xi)\exp(\mp in\xi)d\xi$$
$$\times\int_{-\infty}^{\infty}H_{m+2k}(\alpha y')H_{n+2l}(\alpha y')\exp(-\alpha^2y'^2)dy'$$
$$=\frac{2\pi^{3/2}}{\alpha}[(m+2k)!(n+2l)!2^{m+2k}$$
$$\times 2^{n+2l}\delta_{mn}\delta_{(m+2k)(n+2l)}], \quad (10)$$

where δ is the kronecker delta function. The solution of Eq. (10) applied to Eq. (9) leads to the expansion coefficients

$$C_{m+2k}^{\pm m}(\alpha)=\frac{\alpha}{2\pi^{3/2}}\left(\frac{k!(m+k)!}{(m+2k)!}\right)$$
$$\times\int_{-\pi}^{\pi}\int_{-\infty}^{\infty}g(y',\xi)\exp(\mp im\xi)H_{m+2k}(\alpha y')dy'ds. \quad (11)$$

Equation (9), with the coefficients of Eq. (11), represents the function g by a Gram–Charlier series in the radial direction, and by a Fourier series in the azimuthal direction.

Equation (5), with the coefficients of Eq. (11), becomes

$$f(x,y)=\frac{\alpha}{\pi^{3/2}}\sum_{m=0}^{\infty}\sum_{k=0}^{\infty}\epsilon_m$$
$$\times\frac{[k!(m+k)!]^{1/2}}{(m+2k)!}\exp[-(\alpha^2x^2+\alpha^2y^2)]$$
$$\times\mathrm{Re}\left(\int_{-\pi}^{\pi}\int_{-\infty}^{\infty}g(y',\xi)\exp(-im\xi)\right.$$
$$\left.\times H_{m+2k}(\alpha y')dy'd\xi U_{m+2k}^{m}(\alpha x,\alpha y)\right) \quad (12)$$

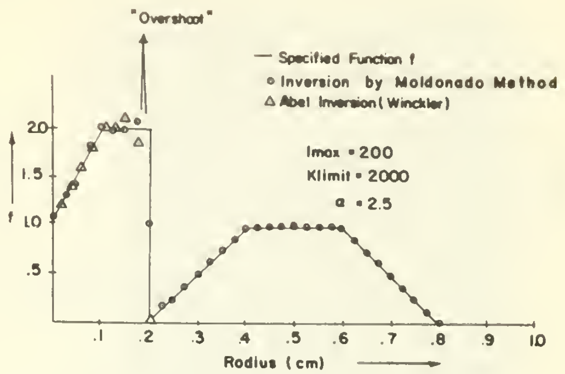


FIG. 3. Axisymmetric test case; Winckler's test.

or, inserting Eq. (6),

$$f(x,y)=\left(\frac{\alpha}{\pi}\right)^2\sum_{m=0}^{\infty}\sum_{k=0}^{\infty}\epsilon_m\frac{(-1)^kk!}{(m+2k)!}$$
$$\times(\alpha^2x^2+\alpha^2y^2)^{m/2}L_k^m(\alpha^2x^2+\alpha^2y^2)$$
$$\times[B_{m+2k}^{m}(\alpha)\cos(m\phi)+D_{m+2k}^{m}(\alpha)\sin(m\phi)]$$
$$\times\exp[-(\alpha^2x^2+\alpha^2y^2)], \quad (13)$$

where

$$B_{m+2k}^{m}(\alpha)$$
$$=\int_{-\pi}^{\pi}\int_{-\infty}^{\infty}g(y',\xi)\cos(m\xi)H_{m+2k}(\alpha y')dy'd\xi, \quad (14)$$

$$D_{m+2k}^{m}(\alpha)$$
$$=\int_{-\pi}^{\pi}\int_{-\infty}^{\infty}g(y',\xi)\sin(m\xi)H_{m+2k}(\alpha y')dy'd\xi. \quad (15)$$

Equations (13)–(15) represent the fundamental inversion by Maldonado's method. Both analytical and numerical inversions were demonstrated in his series of three papers.

The method applies quite directly to density fields. The zero value of the function $\Delta\rho$ studied in interferometry is arbitrary and can always be chosen such that the function is zero outside of a given circular boundary. Singularities do not occur in real density fields except for the spaces occupied by opaque objects. Cases with simple solid objects appear amenable to this method, but will not be discussed here. Shock waves present no analytical difficulties, although they require a high number of series terms for representation.

B. Numerical Procedure

Since experimental data is normally not available in analytical form it was necessary to develop a numerical procedure to apply to the basic integral inversion equations. Although a computer program had been presented by Olsen[8] it was thought simpler to reprogram the equations. This allowed the inclusion of certain features which were useful in the reduction of our data. A more complete description of the computer program is given in Matulka.[9] This includes a detailed discussion

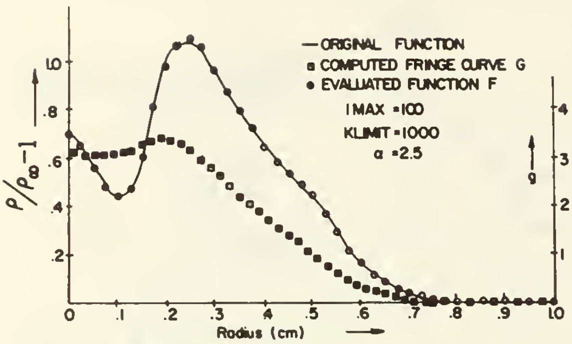


Fig. 4. Axisymmetric test case; free jet flow #1.

of shock waves, symmetry effects and the use of add-on functions.

The effectiveness of the inversion method is shown in the next series of figures.

Winckler, in an extensive application of the Abel inversion method to free jets,[2] used a hypothetical test function that is shown in Fig. 3. Plots of the solution he obtained are shown for comparison with that obtained by the method of Maldonado. The characteristic "overshoot" of the Abel method near a shock wave discontinuity is shown, at this same shock wave discontinuity the largest error by the Maldonado inversion was 3.8%.

Figures 4 and 5 are inversions of typical axisymmetric density functions from the Winckler analysis of a free jet. They demonstrate the capability of the Maldonado inversion routine with realistic types of functions. Both inversions were accurate to within 2.6%.

Figure 6 represents three cross sections of an asymmetric test case. The function is an elliptical cone of base diameters 0.7×0.5 cm, centered at $x=0.0707$, $y=0.0707$ cm. In the solution, for $\xi=45°$ the tip of the cone is rounded by the natural smoothing characteristics of the inversion method, but otherwise the maximum error is 1.5%. This test case represents a complete test of the inversion procedure since it involves both the $B_{m+k}{}^{m}$ coefficients (14) and the $D_{m+2k}{}^{m}$ coefficients (15).

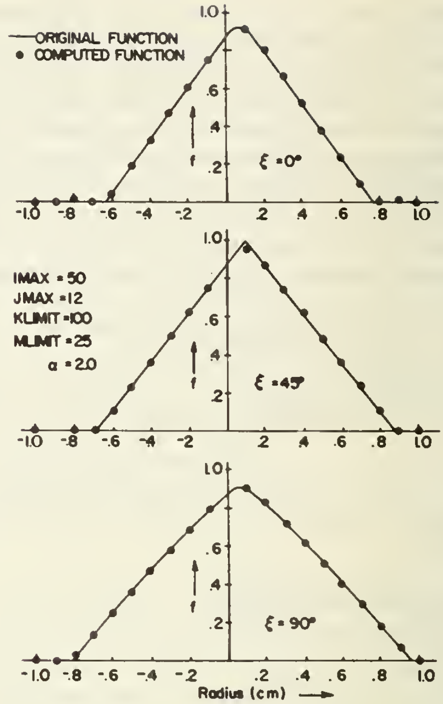


Fig. 6. Asymmetric test case; elliptical cone.

III. EXPERIMENTAL APPROACH

A. Description of the Apparatus

1. *The Holographic Table*

A holographic work table was designed and constructed to provide optical bench facilities about a free jet, as shown in Fig. 7. The table was designed to evaluate various arrangements of optical components to achieve a wide angle of holographic field of view, and to suppress the vibration caused by the free jet noise.

The table was made of pressed plywood laminate, two and one-half inches thick. The four by six foot table was mounted on a rotatable set of plywood boards, with a centerhole of 8-in. diam to accommodate

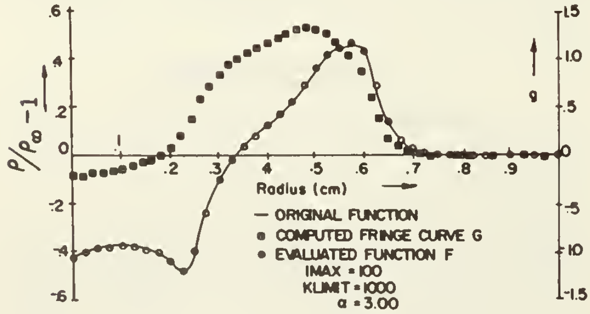


Fig. 5. Axisymmetric test case; free jet flow #2.

Fig. 7. Photograph of the holographic table.

the free jet. This arrangement provided the capability of rotating the entire table in its own plane about the flow field. Below the rotating arrangement, the table rested on four small tire tubes to provide structural vibration isolation from the building. For experiments conducted in C-W gas laser holography with up to thirty second exposure times (without the free jet) the arrangement was very successful. Beneath the inner-tube mounting, the table rested on four standard automobile-type screw jacks. Recessed jack points allowed the table to be readily tilted to about 15° about the flow field.

A Korad K-1 pulsed ruby laser operating at a wavelength of 6943 Å was employed with a Pockels cell Q switching device. The resultant effective hologram exposure time was about 20 nsec, eliminating problems with vibration duiing the hologram exposure. There does remain the problem of vibratory misalignment of the optical components between the two exposures of the holographic interferogram. To help damp accoustically-caused vibration, the mirrors were all mounted on heavy metal blocks. The weakest link in the setup appeared to be the beam splitter holders. They were lighest of the table components, and as a result of their vibration, holographic interferograms obtained tended to have a finite fringe. Occasionally a hologram would be unusable because the fringe spacing became too fine to resolve.

2. *The Test Section*

The table was mounted around a standing free jet. The plenum chamber was about 45-cm long by 30-cm diam. The jet extended 45-cm above the plenum chamber with an inside diameter of 3.18 cm and a throat diameter of 2.0 cm at the exit. The test section was defined by a square plexiglass enclosure, the four inside surfaces of which were inscribed with a 1-cm² grid. The grid box provided some vibration insulation for the jet, and also produced a self-contained coordinate system for the hologram and the corresponding photographs. The grid box is aligned with the plane of the table and rotates with the table. The holograms were arranged about the grid box as shown in Fig 8, with the coordinate system established as shown. Commercial ground glass plate was used as the diffuser.

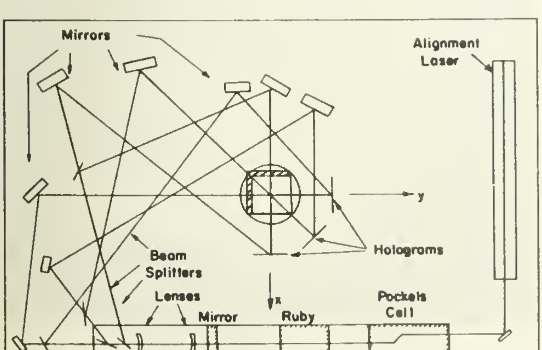


FIG. 8. Schematic arrangement of holographic table.

FIG. 9. Direct print of a hologram (dark field) showing shadowgraph of a free jet at 35 psig.

3. *Laboratory Techniques*

Alignment of the laser beam with the optical components was accomplished by aligning a continuous wave helium-neon gas laser through the rear mirror and along the ruby axis of the pulsed laser. Relative intensities of reference to object beams of about 4:1 were found to yield good holograms.

Since the arrangement of the several mirrors, beamsplitters, etc., so as to have the same optical path length from source to hologram for each beam is a tedious procedure, a cork pinboard was designed to facilitate the table arrangements. Threads of the same length, each one representing a laser beam, were stretched from the laser source to their various holograms via different routes over a sketch of the table and held in position with pins. The table has a six inch grid painted on its surface to facilitate location of the components from the grid on the sketch.

Holograms were made on Agfa-Gaevert 8E75 holographic plates. Normal reconstructions were made with the continuous wave gas laser at 6328 Å. The resulting image magnification from reconstruction at a different wavelength was not considered deleterious. The technique used in making photographs for data extraction will be discussed in the next section.

B. Holographic Experimental Results

1. *Free Jet Experimentation*

The Korad giant pulse laser was installed on the table and holograms taken of the free jet exhausting to the atmosphere. Figure 9 shows a shadowgram taken at 35 psig plenum pressure. A shadowgram is produced directly on the hologram plate when the holographic image is recorded by a single exposure of the dark field technique (with the ground-glass diffuser absent).

FIG. 10. Axisymmetric free jet, 60 psig $Z = 3.0$ cm.

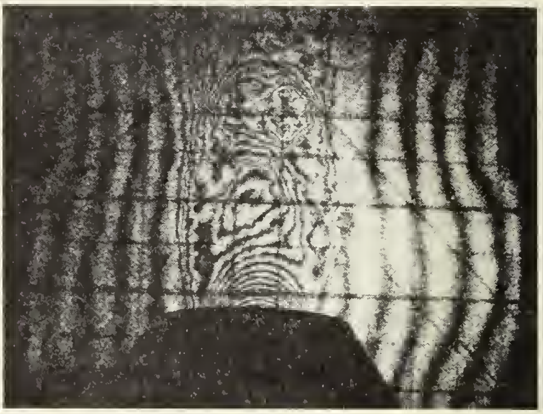


FIG. 12. Asymmetric section of a free jet, 60 psig $Z = 0.5$ cm, 11° tilt, $\xi = 5°$.

Figures 10 and 11 show two views obtained from the same holographic interferogram of jet flow at 60 psig. The interferometric data from this hologram were inverted to provide an axisymmetric solution. Holographic interferograms were also taken at 25, 40, 45, and 50 psig, but were not reduced. Turbulent flow perturbations became more predominant as the pressure was reduced.

To provide a nonaxisymmetric test of the method, the table was tilted 11° clockwise about the y axis of the table. As a result of the tilt, cross sections parallel to the plane of the table through the field became planar symmetric about the x–z plane. The solution of planar symmetric fields require a 90° field of view. Three simultaneous holographic interferograms were taken about the tilted jet at 60 psig with the arrangement shown in Fig. 8. Each of the three holograms provided a field of view of about 15°, one of which had several degrees obscured by the corner of the box. To provide more complete coverage, the table was rotated and additional holograms taken. Two rotations were required. Figures 12 and 13 show the interferograms taken at 5° and 85° which were used in the data reduction.

2. *Experimental Techniques and Considerations*

Previous work at this laboratory[10] had shown the intensity transmission of collimated laser light beyond commercial ground glass falls below 30% of the incident intensity beyond viewing angles of about ±8°. Since the diffraction capability of a holographic plate exceeds ±8°, the ground glass represents the limiting factor to the field of view. In fact, usable holographic interferograms were obtainable with from ±5° to ±10° field of view, centered about the object beam direction, the field size being a function of the ratio of the intensities of object beam and reference beam.

The holographic interferogram appears as a completely three-dimensional set of fringes to the observer. Only special cases of the fringe pattern can be localized, those corresponding to Young's fringes (the finite fringe background pattern), or those corresponding to regions of spherical or cylindrical density variations. To obtain usable data from the fringe number function, one must sample discrete segments of the information. This is done by recording a series of photographic

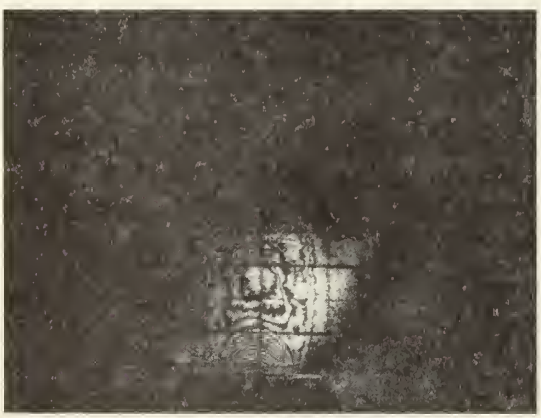


FIG. 11. Axisymmetric free jet, 60 psig $Z = 1.0$ cm.

FIG. 13. Asymmetric section of a free jet, 60 psig $Z = 0.5$ cm, 11° tilt, $\xi = 85°$.

interferograms at regular angles about the test section. For a completely general field one requires 180° field of view, while the planar symmetric case studied required 90°.

There are two basic methods of obtaining sufficient angular coverage of the field. The first is to take several individual holograms, rotating the relative angle between the hologram setup and the test section for each hologram. Unsteadiness in the flow between the exposures will introduce errors. The second method involves arranging a series of holograms about the test section for simultaneous exposure. Gaps in the data from the second method are filled by interpolation. Interpolation over large angles requires that the function vary slowly in the angular direction. This experiment utilized a combination of both methods.

Photographic technique. A normal photograph of the hologram records an image of the focus plane as shown in Fig. 14. Each position on the photograph represents the line of sight from the image through the aperture. All of these lines of sight will represent a nonparallel

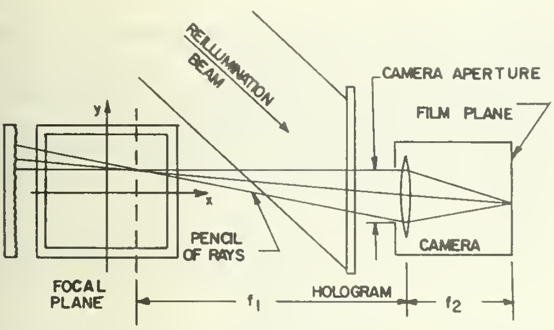


Fig. 14. Effect of aperture size and focus plane position on pencil size of rays about a line-of-sight recorded by camera.

set of lines. For reasonable camera focus distances, the deviation from parallelism is small and may be neglected. The spatial filtering technique shown in Fig. 15 allows the selection of parallel sets of lines of sight for the recorded fringe pattern. The aperture stop at the focal plane of the lens filters out all but the lines parallel to the central angle. The resulting photographs are simpler to analyze since the angles are constant. In addition, fringe data from any z plane may be obtained from the single photograph. Mach–Zehnder interferograms, because of their single collimated source, provide the same type of interferogram.

The technique utilized for this investigation was an application of the lensless focusing capability of the hologram and is easier to achieve than the previous two methods. Figure 16 shows a hologram being reilluminated by a conjugate reference beam of small diameter. Because the reconstruction beam is of small size, the illuminated portion of the hologram represents a small aperture. The resulting image has a large depth of field and photographic film placed at the image

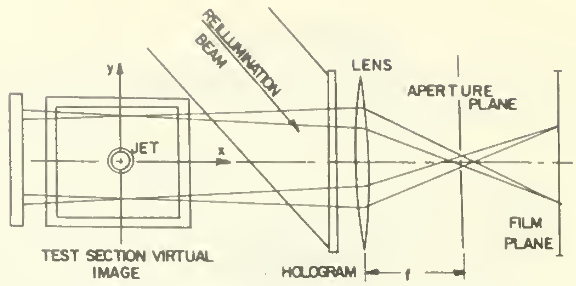


Fig. 15. Spatial filtering technique for selecting photograph of constant angle lines of sight.

records the test scene. Additionally the rays may be focused in the most desirable plane by positioning the film plane, usually near the center of the disturbance. The lines of sight recorded represent the diverging bundle passing through the aperture position from the diffuser. The maximum angle of divergence at the edge of the test field encountered was $\pm 5°$. For resolution greater than $\pm 5°$, one must compensate for the variation. A subroutine of the computer program was written to accomplish this compensation, but the errors introduced by neglecting the bundle divergence have been acceptably small, and use of the routine has not been required.

The interference of coherent light from a diffuse surface causes the photograph to be covered with laser speckle that is inversely proportional to recording aperture size.[11] Although recording apertures of one millimeter were used, the speckle created no problems.

Data reduction. To obtain the photographic interferograms, a camera back with viewing screen was placed in the position of the real image of the hologram as previously described (Fig. 16). The illumination of each position on the hologram corresponds to a particular effective aperture position. The hologram was positioned such that the desired set of front and rear grid lines lined up on the camera back viewing screen corresponding to the desired elevation and angle of view for the picture. Reference to Figs. 10 and 11 show grid alignments at $z=3$ cm, $\xi=0°$, and $z=1$ cm, $\xi=0°$ respectively. Polaroid P/N 55 film was used to record the image.

To obtain the graphical plot of the fringe number function, the negative was used in a photo enlarger to

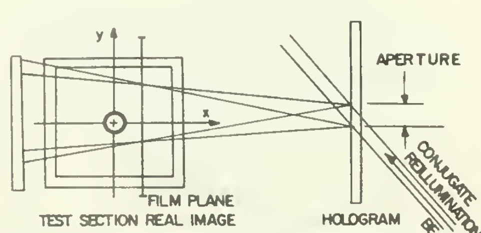


Fig. 16. Lenseless photographic technique using conjugate reillumination beam of small diameter.

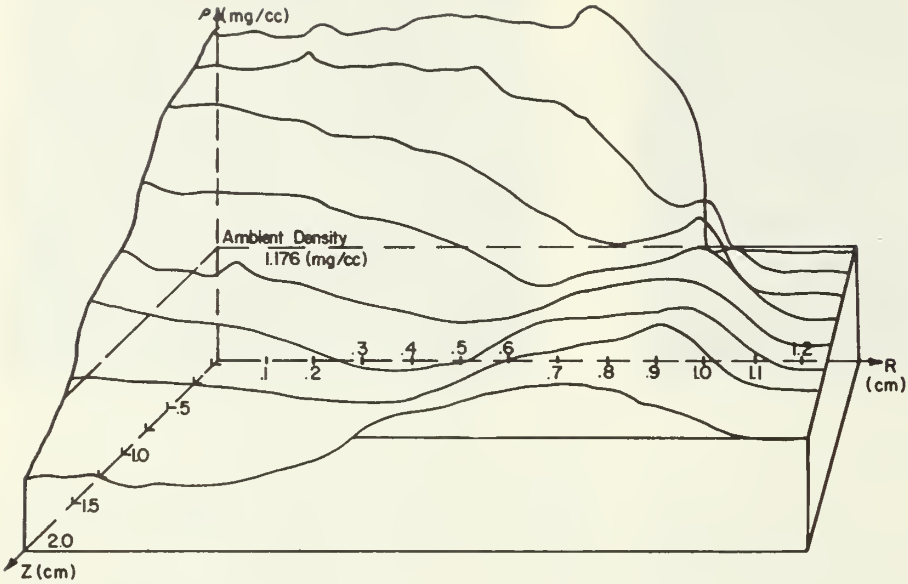


Fig. 17. Topographical plot of the axisymmetric density solution of a free jet at 60 psig.

image directly upon graph paper. With the negative adjusted to match the scale of the paper, the positions of the maxima and minima of the fringes along the desired cross section were determined visually. The fringes were counted, and given a graphical elevation according to their number. Initial fringe numbering was arbitrary, commencing with a fringe well to one side of the field and proceeding across the field, the light regions representing integer values of the fringe number.

For the axisymmetric case the entire procedure was repeated for each level z plane to be solved. Since each aperture position on the hologram corresponds to a particular elevation and angle of view, a new picture was made for each horizontal z plane through the field.

For the asymmetric case, a new picture was taken for each angle ξ desired. When the angle desired was

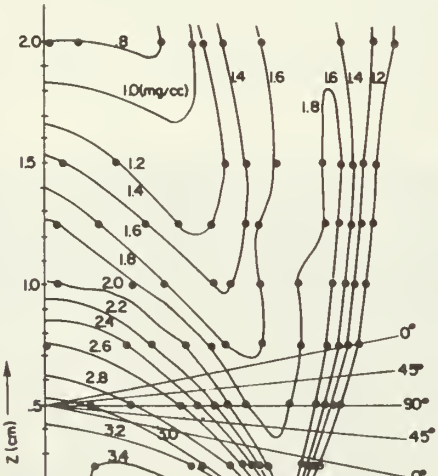


Fig. 18. Isodensity line plot of the axisymmetric solution, 60 psig.

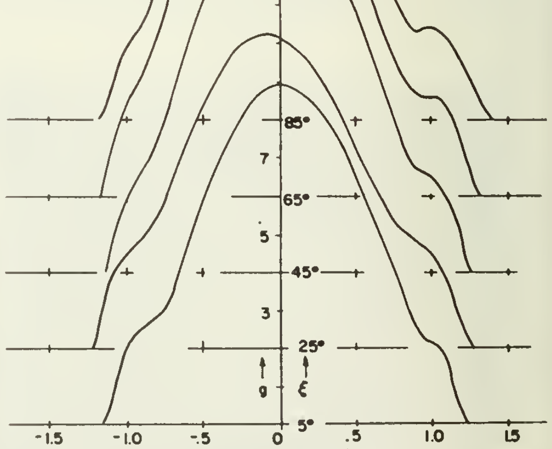
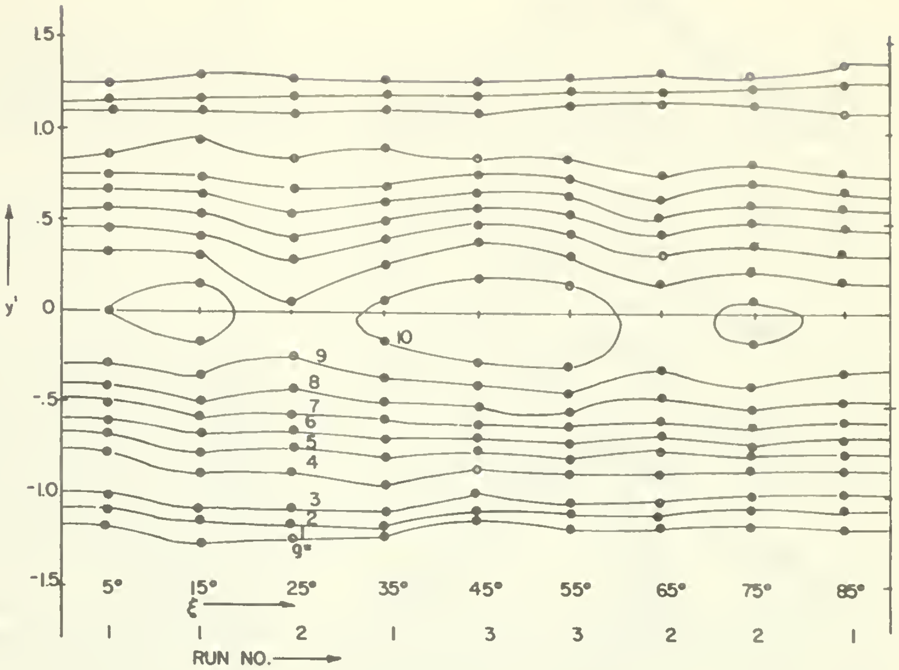


Fig. 19. Fringe number G curves obtained for the 11° tilt free jet at 60 psig.



IG. 20. Contour map of the function G, 60 psig, 11° tilt.

not available because of gaps in the holographic coverage, the curve was graphically interpolated from nearby curves on either side. Fringe curves were obtained in the standard manner for angles on each side of the missing angle. The two curves were then overlayed on a light table and an intermediate curve drawn at the proper relative distance.

IV. THE APPLICATION OF THE INTEGRAL INVERSION TECHNIQUE TO THE EXPERIMENTAL RESULTS

A. Axisymmetric Solution

The interferometric data from the hologram of the free jet at 60 psig were reduced at eight positions out to $z=2.0$ cm in the manner described previously. The complete set of solutions are shown topologically in Fig. 17. The solution compares quite well with the topological features of the free jet solutions obtained by Ladenberg, Van Voorhis, and Winckler[12] in their very extensive interferometric analysis of free jets of one centimeter diameter. Figure 18 is an isodensity line plot of the obtained data. Qualitatively, the densities compare very well with the similar plot of the Winckler solution. Exhaust density in the central region of the exit plane agrees within 0.2 mg/cc (5.3% of the maximum field density) and in the central region at $z/d=1$. (d=nozzle diameter) within about 0.1 mg/cc (2.8%), rising to a maximum of 0.4 mg/cc (10.6%) between. At a relative radius of one-half, the difference in the two solutions runs from about 0.05 mg/cc (1.4%) at $z/d=0$, rising to about 0.2 mg/cc at $z/d=0.25$ and falling to 0.1 mg/cc at $z/d=1$. At the jet radius, the maximum differences fall to about 0.05 mg/cc.

B. Nonaxisymmetric Solution

Figure 19 shows the measured fringe curves obtained for the free jet at 60 psig and 11 deg of tilt, on a plane parallel to the table which intersects the jet axis at a

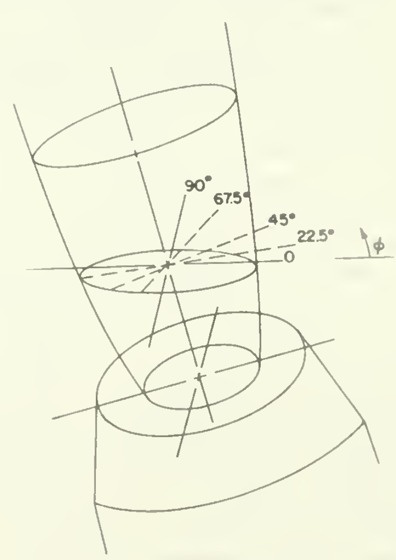


FIG. 21. Sketch of tilted plane showing solution lines.

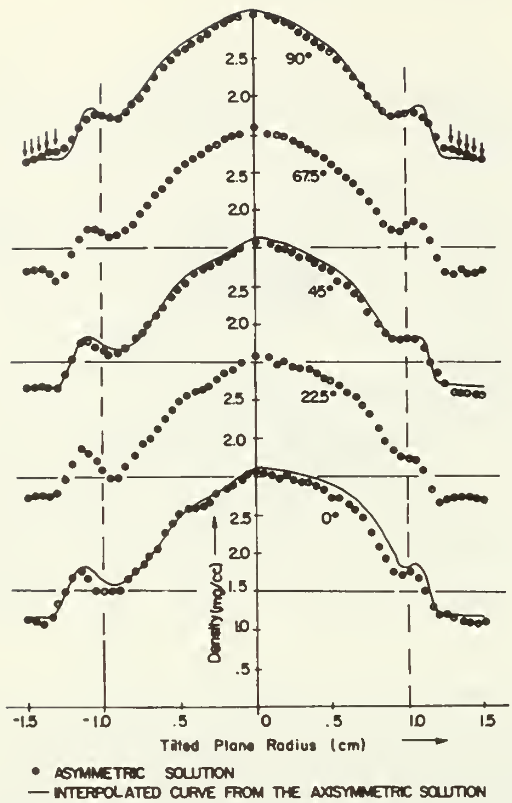


FIG. 22. Solution of the tilted plane density on five diameter lines, 60 psig.

point 0.5 cm from the nozzle. A total of nine angular positions were sampled, every other one being shown in the figure. The significant trend in these data is the shoulder increasing with decreasing angle on the left-hand side and with increasing angle on the right-hand side. A contour map of the data surface from $\xi=0°$ to $\xi=90°$ is shown in Fig. 20. Since not all data could be obtained simultaneously, the run number from which the holograms used were obtained is shown. The nonregularities appear to be the effects of errors introduced in the correlation of angular views taken at different times. Figure 21 shows a sketch of the tilted plane through the jet, with the five lines shown along which solutions were obtained. Figure 22 shows the solution at the five diametric cross sections from $\xi=0°$ to $\xi=90°$. The effects of the shoulder variation in the input data show up here as variations in the position and size of the density "valley" which one can see in the axisymmetric topological plot of Fig. 17. The comparison of the solution from the tilted plane at $\xi=0°$, 45, and 90° is made with the solutions taken from the axisymmetric experiment in common plots of the two functions. The shoulder and valley features of the two solutions appear consistent. The central "hill," is consistent in the two solutions, to the point of showing a common inflection in the slope near radius ±0.5. The outer five points on either side of the 90° curve include convergence errors arising from failure of their series evaluations to converge. The maximum difference in the two solutions is about 8%, although the mean deviation is much less.

C. Discussion of Errors

1. *Numerical Inversion Errors*

Several function shapes were investigated with the axisymmetric inversion to determine the effect of inversion parameters upon the accuracy of the inversion. Errors resulted primarily from the failure of the series evaluation to converge within the maximum number of terms for which computed coefficients were specified and from the approximation caused by representation of the continuous function g by a discrete set of values.

Convergence of the series evaluation is fastest in regions where the functional shape matches a gaussian. As few as five terms in the series were sufficient to evaluate a gaussian function to within 1% everywhere. An axymmetric test case of a displaced gaussian (used by Maldonado) used only a minimum of 25 terms, to achieve accuracy to within 0.8%. The opposite extreme occurs when functions with steep gradients or discontinuities are inverted. For example, in the test of Winckler's function (Fig. 3), the region near the discontinuity required up to 1700 terms for convergence. For functions with such discontinuity, or for experimental data inversions where there are normal irregularities in the data, the center position (at radius zero) convergence is very slow. Interpolation of the center value from nearby points normally reduces the error at the center to within two percent.

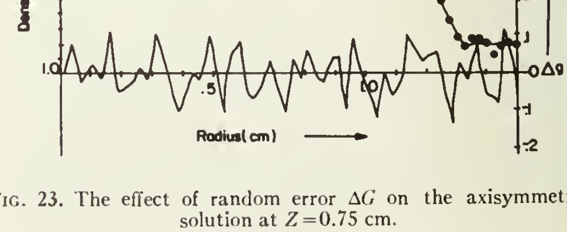


FIG. 23. The effect of random error ΔG on the axisymmetric solution at $Z=0.75$ cm.

2. *Errors in the Data*

Probably the greatest source of error in the data arises from the unsteadiness of the jet flow. The three regions of irregularity in the solution of Fig. 18 correspond to 0.1 and 0.2 mg/cc respectively. Based upon the maximum density in the field, these correspond to 2.8% and 5.6% errors. Variations in the flow between the runs are considered responsible for the major fluctuations evident on the data plane shown in Fig. 20. The asymmetric solution tends to spread errors over the whole field, reducing their effect by statistical averaging.

Graphical positioning of the interferogram and scale matching in the photoenlarger to graph paper step is accurate to within 2%. However, a 2% position error might be magnified to a 5% density error in regions of normally high density gradient. The accuracy to which one can determine the fringe number position in reading the interferogram is within $\pm\frac{1}{8}$ of a fringe, or within about 1.5% of the maximum fringe number of the flow interferograms studied. Winckler has provided an analysis of the relative merits in reading several different fringe arrangements to minimize the fringe number error. Figure 23 shows the inverted solution of the axisymmetric test case at $z=0.75$ cm with a second solution superimposed. The second solution has been made from data with added random error of 0.0625 standard deviation, corresponding to $\pm\frac{1}{8}$ fringe number error in reading the data. The resulting error varies with a maximum of 2.1%.

The background finite fringe spacing is assumed to be constant through the field. An error curve that starts at zero at the boundaries of the flow and rises to one-half fringe in the number would yield a maximum solution error of 0.18 mg/cc at the center, or about 5.0% for the axisymmetric solution.

It is estimated in the solution presented herein that probable errors are less than 8% everywhere, corresponding to the maximum difference in comparison of the two solutions in Fig. 22.

V. CONCLUSIONS

A self-consistent demonstration has been made of an interferometric method for the acquisition of data about a three-dimensional density field of one plane of symmetry. The data set has been successfully inverted by the application of a recently developed mathematical inversion scheme and shown to be reliable to within eight percent of the density range. The mathematical model has been tested for realistic density patterns, representing supersonic wakes and jets. The pulsed laser method of holographic interferometry can be successfully applied in environments relatively hostile to normal interferometry. There are no inherent restrictions to the application of the method to generally asymmetric fields. The optical arrangement of the system is highly flexible and can be modified for interferometric studies of wakes, rockets, turbomachinery flows and other hostile, or highly transient events.

* This work was supported by the Naval Air Systems Command through the Naval Weapons Laboratory, China Lake, Calif.

† Now Research Programs Officer, Office of Naval Research, Arlington, Va. 22217.

[1] C. D. Maldonado and H. N. Olsen, J. Opt. Soc. Amer. 56, 1305 (1966).

[2] J. Winckler, Rev. Sci. Instrum. 19, 307 (1948).

[3] L. O. Heflinger, R. F. Wuerker, and R. E. Brooks, J. Appl. Phys. 37, 642, (1966).

[4] H. W. Liepmann and A. Roshko, *Elements of Gas Dynamics* (Wiley, New York, 1967), p. 165.

[5] C. D. Maldonado, A. P. Caron, and H. N. Olsen, J. Opt. Soc. Amer. 55, 1247 (1965).

[6] H. N. Olsen, C. D. Maldonado, and G. D. Duckworth, J. Quant. Spectry. Radiative Transfer 8, 1419 (1968).

[7] C. D. Maldonado, J. Math. Phys. 6, 1935 (1965).

[8] H. N. Olsen, C. D. Maldonado, G. D. Duckworth, and A. P. Caron, Aerospace Research Lab. Rep. ARL 66-0016, 1966.

[9] Robert D. Matulka, Ph.D. Thesis, Naval Postgraduate School, 1970.

[10] J. B. Sullivan, Master's thesis, Naval Postgraduate School, 1968.

[11] L. H. Tanner, J. Sc. Instrum. 1, 517 (1968).

[12] R. Ladenberg, C. C. Van Voorhis, and J. Winckler, Phys. Rev. 76, 662 (1949).

DISCUSSION

(Question) Is this still a stationary case?

(Answer) This is still stationary, but there is nothing inherent in the system that prevents you from taking multiple holograms and conducting a non-stationary investigation. In the particular case we are dealing with, we did have stationary flow; in order to get more information, we actually rotated the table to get a series of six different holograms.

(Question) In the turbine case are you taking a picture of no flow and then a second picture with flow?

(Answer) I really haven't decided the procedure; in the case of the problems I am talking about today, we had a complete field of view, a complete pass through the system. In a compressor a different procedure might be needed.

(Question) What is your uncertainty in the fringe measurement?

(Answer) It's typical of the order of a 1/10 fringe.

(Question) I think you may have been a little bit hard on yourself with that 5% error since the errors in those two curves were independent. The actual error in either measurement was only about 5%.

(Answer) Yes, that is true. If we had been in a process where we got 20% differences or something like that, then I would have wanted it more accurate in describing the error involved.

(Question) In your jet work you had to have viewing angles of 90 degrees?

(Answer) Yes.

(Question) In a turbomachine can you get multiple viewing angles?

(Answer) I can get multiple viewing angles in the entrance region just before the blades of the rotor, and I can probably get information behind the rotor and in front of the stator. There are a lot of planes available. Professor Vavra is particularly interested in the entrance of his transonic turbine, and we are going to be emphasizing this in our preliminary work. When that is done, we shall go on to look at other points. Perhaps we will be able to look at the tip of the blade, for example.

(Vavra) You mentioned a turbine. Actually it is a transonic compressor with no guide vanes in front of the rotor. What we would like to do is to define the shock patterns in front of the first rotor. It is a standard arrangement for a rotor. We have quite a large amount of room since it is only 11 inches in diameter, and we have a fairly large distance ahead of the rotor.

(Question) I was thinking of the density field around the blades. If you have many blades, there is no view pattern.

(Answer) True. There are 17 blades and so there is viewing angle trouble. People at your place, for example, have shown the problems involved. I think your point is well taken. The viewing angle in turbomachinery is considerably limited, and the question now is what can we do with a considerably limited viewing angle if you go outside of the entrance region. Would you use a double pulsing, for example, as people did at your place, or would you use some other technique? I think the double-pulsing technique is probably a very productive way to go.

(Question) You didn't mention that there are many other applications in turbomachinery for the laser, including obtaining the motion of the blades.

(Answer) Yes, there are many more applications of holography that we have not reviewed here. A recent conference in France has indicated many more engineering applications of holography.

APPLICABILITY OF CASCADE TEST DATA TO DESIGN METHODS

Discussion Leader: Mr. R. E. Olson

DISCUSSION

(Olson) I guess I have the unpleasant task of trying to get a discussion going on the applicability of what Prof. Vavra has called the classical two-dimensional design techniques. Later on we shall get into the three-dimensional approaches or maybe what we might do to get away from such approaches, if that is the desired thing to do.

I thought at first, that we should decide in what areas the classical two-dimensional approach is valid, or we feel is workable from an engineering standpoint. Then maybe we could go on from there and discuss what we feel some of the limitations of the classical two-dimensional approach are and how much further these two-dimensional approaches can be taken. I have a few thoughts of my own, but much of it I am hoping will come from the floor.

Just to start the discussion, I would say that it probably is generally concluded that if you have two-dimensional or axisymmetric flow coming into a stage which is, let's say, a constant annulus stage with fairly thin approaching boundary layers, low tip speeds, subsonic, we could all agree that the two-dimensional design procedures which make use of some kind of axisymmetric equations of motion, including the radial-equilibrium equations, in conjunction with cascade test data for turning and losses - that is the kind of classical two-dimensional procedure I'm talking about - would be applicable for that kind of a case. Is there any agreement there or....

(Papailiou) I think that you have to introduce the loading; that is, if the blades are highly loaded, I don't think that the classical two-dimensional approach is accurate.

(Olson) Yes, OK. I should have also said that the stage is working at minimum loss, which is a requirement.

(Serovy) What do you mean when you say classical two-dimensional methods, I would like to hear just what do you mean? Are you talking about designing an axial-flow compressor or axial-flow turbine?

(Olson) I am talking about writing a set of axisymmetric equations of motion in the meridional plane with the blade forces introduced in a circumferentially averaged fashion. We can then talk about a two-dimensional flow path through the compressor with the cascade data being used to introduce essentially the blade forces, and that is done in various ways depending on who does it. I think it can probably best be thought of as writing the circumferentially averaged equations of motion for the annulus, and then introducing blade forces into those equations of motion through the cascade correlations of turning angle and losses. Maybe somebody else has a better way of looking at it. That's what I visualize.

(Lakshminarayana) We can do better than that. With the actuator disc theory, you can get the solution downstream or upstream and then solve for the blade-to-blade flow.

(Olsen) But that can also be done in the approach I outlined. You can write the equations of motion through the disc; and if you could put in the distributed blade force into those equations of motion, then that would ...

(Lakshminarayana) I believe you are thinking of an analysis similar to what Dr. Smith has done.

(Smith) I have been sitting here wondering whether I should start to talk now or wait a little while, because I am scheduled to be a discussion leader later on in the program. Perhaps, Professor Vavra, you would agree that it would make more sense to discuss calculation procedures for both compressors and turbines at this session rather than try to take them up separately.

(Vavra) Yes, I would agree.

(Smith) Well, then, I would say that what Dr. Lakshminarayana is referring to should be considered to be one of the elements in Professor Vavra's so-called classical approach where we essentially, by taking circumferential averages, arrive at a two-dimensional problem, which is tractable these days with computers.

In arriving at the circumferential-average equations in a fairly rigorous way, you find that there are some second-order terms which are left over. I made an attempt to evaluate the magnitudes of the second-order terms for one case and found that they were not particularly large and that neglecting them would be appropriate. The case that I looked at treated blade-to-blade velocity distributions as linear variations from one blade to the adjacent blade (an approximation which certainly isn't a good approximation when there are strong shocks in the rotor). (to Schwar) I am waiting to see what you have to say about that. I see that you are on the agenda to talk about radial equilibrium across normal shocks. It's quite likely that, with such strong non-axisymmetric disturbances in the flow, the circumferential-averaging procedure that leads to the axisymmetric equations can no longer be a very good approximation.

(Olsen) Now, while we are talking about non-axisymmetric effects, I think that loading comes in here. What about high loading? How do you feel the applicability of that kind of approach carries to the high-loading case where you might have significant secondary flows or separation?

(Smith) I would say that the loading can enter in two ways. First, the approximation that shows that the second-order terms are not too important breaks down as the loading gets high; in other words, the second-order terms get more and more important as the lift coefficients on the airfoils get larger and larger, even without consideration of viscous effects. Second, with consideration of viscous effects you get something pretty close to chaos if you really have an over rotor or stator. The flow can be highly three-dimensional; these are situations we try and avoid but which you can't always completely avoid, of course. I think that the classical approach for that kind of case is no good.

(Olsen) So this is where we possibly could identify a limitation of the classical approaches.

(Smith) Yes. It is a band rather than a sharp line, I think, because there is no clear distinction between being not overloaded and overloaded; but there is an area where the approximation gets worse and worse as you load higher and higher.

(Unidentified) I am rather confused about the definition of the classical 2-D approach. What I understand now is that we are solving the equation of continuity and energy using cascade velocities and simple radial equilibrium. I mean we are solving that, but are we including streamline curvature or not in the 2-D approach?

(Olsen) I think including streamline curvature still falls into the category of the classical two-dimensional approach. I think that there are various degrees of sophistication that we can still categorize as the classical two-dimensional approach.

(Unidentified) Then it would include streamline curvature.

(Smith) Let's say including streamline curvature, including components of blade forces which are either in radial direction or in all directions, but essentially doing only what is necessary to reduce the problem to a two-dimensional solution and getting as complicated as you like within that framework.

(Olsen) This is not limited to an infinite-span assumption,which is not a real case.

(Smith) Certainly not limited to infinite span.

(Olsen) This is, I think, one thing that we should talk about; what is the influence of the wall boundary layers, how are these included in the analysis? I think, maybe we should talk about cases where you have either physical convergence, the annulus is converging, or talk about convergence due to the presence of boundary layer. These are the kind of things which we have to consider with regard to real machines, and that is what limits us.

(Yampolsky) Aren't we talking about - that's what Dr. Smith started talking about - the assumption of axial symmetry that brings you from the three-dimensional case to the two-dimensional case? And it is a question of whether you are playing with a small correction to it or are you going to look for a different rule of adding energy. In part you make the assumption to make the mathematics more tractable, but at the same time it confines you to some energy-addition scheme and a scheme for the vorticity and a circumferentially averaging out of the circumferential component of vorticity. So really you have an underdetermined problem, and I don't know if you have any physics to tell you to do any different.

(Smith) Perhaps it is nit-picking, but I object to starting with the assumption of axial symmetry and presuming blade forces and arriving at the equations. I think that is the wrong way to think about the problem. You should start with the general three-dimensional equations, take circumferential averages of the three-dimensional equations, and show that these do reduce to the axisymmetric equations.

(Yampolsky) You will have to put something in, though, in terms of how do you add the energy and you have a reason for adding it.

(Smith) The energy is not added by some mysterious hypothetical distributed blade force but through the usual unsteady equation that is necessary to add energy:

$$\frac{DH}{Dt} = \frac{1}{\rho}\frac{\partial P}{\partial t}$$

H = stagnation enthalpy
t = time
ρ = static density
p = static pressure

The distributed blade force, if you still want to call it that, falls out by taking circumferential averages of the equation.

(Yampolsky) Yes, but somebody said yesterday that part of the problem was to be able to go from stall to choke. Now, ignoring how you arrive at the design problem and how you get the blade shape, suppose that you can, with an infinite number of experiments and a very clever set of development people bending and twisting blades, arrive at a blade surface which is the plane orthogonal to the absolute vorticity and the relative velocity vector fields. Now you have this blading and the flow matched at design point. It is immaterial whether you did it analytically or with a development program. You want to go from the design point towards stall so that this blade, which doesn't change to the flow as you would like it to do, is mismatched and that is finally when you fall off the cliff, at stall. The flow is trying to keep itself together as you go in the opposite direction, so if you want to predict you will almost have to start from this surface. So I don't know whether the argument you make of averaging back circumferentially and calculating it that way gives you the clue towards what's probably happening at stall.

(Smith) Of course you are getting into the cascade questions there, and the distribution of loading along the chord and from hub to tip is likely to be quite strange when you are far removed from a good design point. But we are talking really concepts and calculations now, and there isn't any reason why that can't be included in the model if you strive to do that. You can still talk about a through-flow two-dimensional model and a blade-to-blade two-dimensional model which, when added toegther, give you a quasi-three-dimensional picture. It certainly is appropriate, I think, if we are trying to calculate the flow accurately for conditions far removed from design, to include in the through-flow model local regions of high loss in the blades and, if we are clever enough, local regions of flow separation that can be introduced in the form of circumferential thickness blockage. Normally we allow for circumferential thickness blockage for the metal and for the boundary layer on the blades. This boundary layer can be very large if you want to incorporate separation. So, I think these things can be done within the framework of this so-called classical approach. I don't know what depth various investigators have tried to pursue this. We've gone so far but not all the way. I guess you people have too.

(Bullock) Dr. Smith's remarks are very relevant. His justification of our quasi-three-dimensional calculation techniques is the best I have read. We all recognize that the physical limitations of our calculation techniques require us to make very broad assumptions about the flow. No matter how reasonable or unreasonable some of these assumptions appear to be, we accept them because they yield the best competitive turbomachinery.

Until recently, experiments and theoretical analysis of two-dimensional cascades have been our main sources of data and understanding. I believe, however, that this well of information is being exhausted. Very plausible correlations of the two-dimensional losses and deviation angles have been achieved. When we apply them to actual turbomachinery, however, we have to make arbitrary corrections; the corrections vary from individual to individual, based on his personal experience. In general, the available correlations for axial compressors are quite unsatisfactory.

A number of important questions can not be answered by research with two-dimensional cascades:

What is the pattern of the shocks in transonic rotors or stators? Is it even possible to have a continuous shock surface through which the flow is deflected in such a manner that radial and circumferential equilibrium are maintained? Is time-steady flow possible even under ideal conditions?

What happens when the radial velocities are not negligible, and when noticeable changes in axial velocity take place? Some aspects of the circulation developed by the blade are certainly altered, and deviation angles and losses would be affected.

What are the effects of rotating flow? Our quasi-three-dimensional techniques partially treat it. The question about how to calculate the flow through shock waves was raised above, but what about the boundary layers? They move radially outwards or radially inwards, depending on whether the airfoils rotate faster or slower than the flow. Data exist which show that this migration causes turbo-airfoils to behave differently than they do in two-dimensional cascades. Measurements behind midspan dampers and behind fan splitter vanes support the notion that the radial migration of boundary layers is of more than casual significance.

At the hub and casings of turbomachinery, the boundary layer is even more complex. As in two-dimensional cascades (without suction on the end walls), part of the boundary layer flow at the blade ends rolls up near the suction surface, and proceeds on its merry way without any apparent regard to what the rest of the fluid is doing. When the blade ends are unshrouded, the picture is further confused by flow through the clearance at the blade ends. Shrouded blades have flow through the shroud seals that blur our comprehension. Moreover, the incidence angle varies rapidly with radius at the blade ends, and the limiting angle of incidence can be quite different than what we think it should be. Although we have virtually no quantitative understanding of the fluid mechanics near the blade ends, it is these very regions that limit the output of a blade row and control its overall efficacy.

We finally have to consider unsteady flow. Some concepts developed for airplanes have been partially translated for turbomachinery. Anyone who believes that we can build a good model on these ideas should recall the motion pictures made by Kofskey and Allen and reported in NACA TN 3260.

Additional problems appear, however. The time-unsteady rotating stalls have both aerodynamic and mechanical significance. Of probably greater import is the fact that stall itself is the heir of a time-unsteady situation. Anyone who has had the opportunity of observing an airfoil go into stall is aware of the fact that separation is initiated by a stagnant cell near the trailing edge. The stagnation point caused by the cell forces the flow ahead of the cell to decelerate (at the same time, the cell causes the adjacent flow along the span to accelerate). The stagnant cell moves upstream; and if the spanwise relief is insufficient, the cell grows and continues to move upstream until a vital portion of the flow is disrupted.

The same events undoubtedly occur in a blade row. Evidence suggests that the initial disturbance starts on one blade. When this blade is thoroughly stalled, the stagger angle of the blade row causes this stall to propagate circumferentially, and the familiar rotating stall develops.

One can argue that the suppression of an initial stall zone could delay the stall of an airfoil or diffuser. Circumferential slots in a conical diffuser could accomplish this suppression, and there is data to show that such slots do inhibit separation. One may contend that the casing treatment being studied by NASA relieves the high pressures created by a stalled passage, prevents its circumferential propagation, and thus delays rotating stall or surge.

These are the important problems for future work. The two-dimensional cascade does not appear to be a useful device for solving them. More attention should be given to the geometrics afflicted with real maladies, even though the experimentation and analysis are more difficult and more challenging.

(McBride) It seems to me that the fundamental question here is whether or not plane cascade data can be an adequate representation of what happens in a machine. Personally, I have doubts as to how truly the design rules represent even the two-dimensional case since they represent correlations for tests of a very limited and non-representative set of cascade configurations. Carter's rule, for example, was derived from a very few configurations, all of relatively high camber and very little, if any, consideration of axial velocity ratio - "peaking factor". We have to decide whether it is deficiencies of this kind which cause our troubles or whether it is boundary layer centrifuging, blade row interaction, and other radial flows. Our course of design method development will depend on our answer to this question.

(Olson) I think we have come to some understanding of what we are talking about when we refer to the classical two-dimensional approach, or better still, the quasi-three-dimensional approach. Certainly this kind of approach is very workable in certain situations for calculating design point, but as you go off the design point then you begin to question how much mileage there is in this type of approach. So that's maybe one limitation that relates to the applicability of our cascade results, that is, the business of the effect of contraction or axial velocity ratio which can be significant in a compressor for a couple of reasons, one just because of the physical configuration and another because of the presence of the boundary layer. I think it can be said that the situation is that there is not much cascade data available to evaluate whether or not cascade data taken at various contraction ratios and then applied in design procedures will adequately project the effect of axial ratio in the machine. Maybe that is one thing that we can discuss.

(Bullock) Would that be consistent with the group that that type of work be done?

(Olson) Yes.

(Lakshminarayana) We have some data at this moment for moderate axial velocity ratios varying from .9 to 1.1. Heilmann* in Germany did some work in high subsonic flow cascades. But what we need is some data with very high and very low axial velocity ratios.

(Olson) Have we come to a conclusion on that point?

(Lakshminarayana) Must these tests be carried out in rectilinear cascades or annular cascades?

(Olson) Yes, that's a question.

(Yampolsky) Yes, but can you do it on a cascade? You're putting another dimension into the cascade, and it's an additional degree of freedom.

(Olson) Does anyone have any evidence that this works? Has anybody tried to apply cascade data obtained at various axial velocity ratios in these design methods and then checked them against the actual case? Alex published a report at Brussels in which he did this for the supersonic case. We were able to use cascade data at various . . . . maybe you want to talk about it, Alex.

(Mikolajczak) I agree with you on the importance of the axial velocity ratio and I suspect that most manufacturers have realized its importance some time ago. We found that we had to have this kind of information for design. In fact, we have a lot of data for subsonic cascades which remains unpublished for proprietary reasons. In the open literature there is very little data. We also find that axial velocity ratio is a dominating parameter for the design of supersonic blading. If we ignore the three-dimensional convergence, we completely mislead

* W. Heilman, Dr. Ing. Dissertation, Technische Hochschule, Aachen, 1967.

ourselves as was shown in our Brussels paper.* In that paper we compared the performance near the tips of three rotors with the performance of cascades designed to simulate these rotor sections. We found that when we matched the axial velocity ratio, as well as incidence and Mach number, we got very good agreement between cascade and rotor losses and flow turning. So, although I recognize that there are possible problems associated with radial equilibrium, etc., at this point in time these effects do not seem to be dominating.

(Smith) One important consideration in the design of transonic and supersonic rotors is establishing the flow or starting the flow, if you like to say it that way. At lower speeds in the rotor the shocks are pushed out in front of the passages, and you certainly should expect that the flow is highly three-dimensional in a low-radius-ratio rotor under these conditions. Now at high design Mach numbers, of the order of 1.5 and up, it is extremely important to have the minimum throat that you can get by with for purposes of having low loss at the design point. So you have to know how small you can make the throat and still get the flow you want, and I question whether you can determine that from two-dimensional tests. So we have been going away from cascade tests in recent years in my company. And as measurement techniques improve, we can get better measurements actually in a rotating rotor. We have been going in that direction, and it looks like the right thing to me.

(Schwaar) In low-hub-tip ratio, high-Mach-number designs, three-dimensional effects of course are important. However, I believe that the basic flow phenomena in the tip region of a transonic compressor rotor can be to a large extent qualitatively observed in two-dimensional cascades. There is a considerable amount of transonic cascade data at Pratt and Whitney, which in my opinion represents very well the physical character of the flow in the tip region of the transonic blades. How to interpret them and put them into an actual design procedure is another thing.

(Olson) Alex can comment on that too, but I think that some of the comparisons that were made were very close to spill.

(Mikolajczak) The data were compared near spill. We have to recognize, of course, that a supersonic cascade operates at unique incidence

* Mikolajczak, A. A. et al, "Comparison of Performance of Supersonic Blading in Cascade and in Compressor Rotors", Journal of Engineering for Power, Trans. of ASME, January 1971.

for a given inlet Mach number at infinity.

(Smith) Yes, but the rotors we work with certainly have very important operating ranges.

(Mikolajczak) Admittedly, but I maintain that the unique incidence condition is not violated in a rotor. Consider what happens to the Mach number and incidence as we operate a rotor along a constant speed line. The need to satisfy the equations of motion requires that the flow ahead of the rotor readjusts. Both the inlet Mach number and incidence will then change, but still in accordance with the unique incidence criterion. In a cascade of a given geometry, a constant incidence is maintained until spill. At this point the periodicity of the flow is lost and no further information is valid. However, we can test the cascade at a lower Mach number and get the performance at a new unique incidence.

To obtain meaningful data we have to be sure that there is no shock reflection from the walls into the cascade. This requires that we bleed the walls. Incidence and internal geometry have to be calculated rather precisely to avoid a starting problem.

We should not lose sight of why we use cascades. If we were always designing within the range of the existing rotor data, further cascade testing would be superfluous. The reason for the cascade testing is to give us a relatively quick and cheap way of getting information in the regime where we haven't been before. This can be a guide to a new rotor design. We must therefore look at a cascade as a tool, and not as an end in itself.

(Hartmann) Can you enumerate what data you can use? Can you use loss data for that point?

(Mikolajczak) You can get turning, losses, and a check of the flow pattern.

(Hartmann) I think it is safe to say that the cascade is used differently in subsonic and in supersonic flow. In subsonic flow you can develop catalogs covering the complete operating range and store them in computers. A supersonic cascade is used differently. I don't think anybody is trying to catalogue all blade shapes or operating ranges and so on and use it in the same way, but properly used it could be very useful.

(Schwaar) Concerning now the applicability of such cascade data to actual blading design in the supersonic or transonic regime, for example at a rotor tip section, one important question arises. The cascade test usually is a 2-D test with parallel cascade walls. But there is the possibility of setting up a cascade test with non-parallel walls. In this case, the axisymmetric flow calculation with radial equilibrium at inlet and exit of the rotor determines a local annulus area change for the tip section stream sheet, which can be duplicated in the quasi-2-D cascade test. Then you would get a very good idea of what happens, at least in the very important inlet region of the cascade. In an actual design, every effort is spent to prevent the relative Mach number at the shock location to exceed the inlet value. In a 2-D cascade, this is done by aligning the suction side in the entrance region with the incoming flow direction and maintaining a straight suction side segment, at least until the covered cascade region is reached. If now you expect such a blade to work in a similar manner at the tip section of a rotor with converging annulus, then you are missing your design objective. In fact, the acceleration of the axial component in the entrance region causes the relative flow direction to swing toward the axial direction; and if the blade suction side is straight in that region, the flow will be prevented to turn in the relative system, and compression waves thus will be generated on the suction side and will propagate upstream, causing a compensating expansion fan to appear at the cascade leading edge. This results in an additional angle of incidence, thus finally in a lower inlet axial velocity and a loss of mass flow rate. This is, aside from viscous and tip-clearance effects, a very clear case where a 2-D cascade test cannot possibly represent the transonic or supersonic flow conditions at the rotor tip section.

(Olson) That's the exact thing that Alex has pointed out.

(Mikolajczak) To make direct comparison between cascades and rotors, we have to test cascades at the same convergence ratios.

(Smith) We have built supersonic and transonic cascades which have converging sidewalls in order to try and approximate the annulus convergence as closely as we could. Of course, we always found that there is considerable growth of boundary layer through the cascade; and even though we used porous walls all over the place, we never did really succeed in effectively removing end-wall boundary layers as had been done in many low-speed cascades. Do you, in fact, use converging walls?

(Mikolajczak) We have not made an attempt to bleed off the boundary layer, because in a compressor you seldom need a contraction ratio of unity and hardly ever a ratio below unity. We have, however, extrapolated data taken at a number of contraction ratios to get performance at a contraction ratio of unity.

(Smith) Yes, but the annular boundary layer conditions in the compressor are quite different than they are in a cascade. You have a kind of boundary layer in a cascade that doesn't exist in a compressor.

(Mikolajczak) Possibly. Tests run with carbon black placed on the airfoils at their junction with the end-walls have given us an indication of how the stream tube contraction changes locally. Results indicated that there were abrupt changes along the wall. However, in the mid-span of the cascade,we were getting stream tube convergence which was not sensitive to wall geometry changes. We may not, therefore, be able to use cascade data right in the very tip of a fan where the strong local perturbations occur. However, away from that area, streamlines are better behaved;and then the cascade data closely approximates the compressor flow.

(Smith) What kind of area contraction do you get typically through a supersonic cascade if the end walls are parallel, in other words, due to the boundary layer growth?

(Olson) Well, if you plot it against pressure ratio, then for low pressure ratios it is practically zero for some cases, while it gets to 10% for high pressure ratios.

(Schwaar) I assume you are including in the area contraction not only the boundary layer buildup effect on the channel walls but also the effect of blade wakes, which can be stronger than the former in the presence of shock boundary-layer interaction.

(Smith) When annulus contraction is simulated by a buildup of end-wall boundary layers in a cascade, I am concerned about the effects of those boundary layers on the performance of the cascade. I would think, certainly in heavily loaded cases, you would get a different pressure rise and a different loss when your shocks are interacting with an end-wall boundary layer that is of a character quite different from that which which occurs in a rotor.

(Olson) We were concerned about this question also; and as a result, we tried different ways of getting a contraction in a cascade and found that there wasn't really a significant difference in performance. The performance was relatively independent of how you got the contraction.

(Smith) Is that true with the static pressure rise also?

(Olson) That would be for a given static pressure rise and a given contraction; the pressure recovery and turning would be the same.

(Smith) How about the static pressure rise that you get for a given shock pattern? I ask the question that way because I, too, have been able to simulate the kindsof shock patterns in a cascade that we found later existing in a rotor through use of the casing static-pressure crystal measurement technique. But the cascade did not get the static pressure rise that the rotor got, presumably because of the thickening of the end-wall boundary layers.

(Mikolajczak) About two years ago we compared the static pressure ratio at spill from three different tunnels (DFVL Germany, V.K.I. Belgium, and P&WA) in which similar airfoil geometries were tested at comparable inlet conditions. The spill static pressure ratios were substantially different. It became apparent that we had to compare the performance on a consistent basis, namely to mix out the flow to uniform downstream conditions. This was particularly important when the exit gapwise traverses were made close to the cascade trailing edge. The discrepancies between data from these three tunnels essentially disappeared when mixing was introduced. We also found that the mixing correction depends on the airfoil shape that is tested, because the flow at the trailing edge is at high Mach number and is strongly non-uniform gapwise. In a rotor the downstream average flow is measured maybe two chords downstream or thereabouts; by then the non-uniformities have essentially disappeared. Comparing on the mixed-out basis, we get good agreement between rotors and cascades.

Let us come back to shocks. I question whether the shock structure which we infer from the tip casing measurements has quantitative resemblance to the actual shocks in the compressor. The measurements pick up the lambda shocks which occur where the passage shock interacts with the wall boundary layer. What the shock structure is away from the boundary layer is open to question. I wonder whether you had any internal flow visualization of the shock pattern in the rotor, which would allow you to make a direct comparison between cascade and rotor.

(Crouse) Another difference between the flows through a two-dimensional versus a three-dimensional cascade is the boundary layer environment. The equilibrium flow condition in a turbomachine requires a static pressure gradient to balance the centrifugal force of the tangential velocity. In a stator the boundary layer tends to move toward the lower static pressure, which is toward the hub. However, in a rotor the situation is different. Near the surface of the blade, viscous drag increases the tangential velocity of the boundary layer. Since the general pressure gradient is not enough to balance the boundary-layer centrifugal force, the rotor boundary layer tends to transport outward. Flow-visualization studies show that both rotor and stator surface boundary layers break into one vortex core on the surface of the blade. Much of the boundary layer flow is transported to the tip of the rotor and the hub of the stator. These losses, although generated over the blade span, are usually charged to the blade end regions.

A blade row with a damper may show quite high losses in the region of the damper, because the damper blocks the vortex core and causes some of its flow to be spilled off the trailing edge of the blade in that region. These secondary transport flows are probably difficult to put in the cascade model, but their effects should be considered.

(Olson) I guess the only way you can determine whether that's significant or not is by the degree of agreement that you get when you compare the cascade data with rotor data; and I guess you can say, from the report that Alex has written, that good agreement exists at least for the small amount of cases that have been compared so far. The initial indication is, I would say, that rotational effects are small in some cases.

(Mikolajczak) You have brought up a very interesting point. One does get quite a lot of transport of the boundary layer. The tendency is, though, to see it close to the hub or the tip, where the direct application of cascade results would be questionable anyway.

(Serovy) In your paper at Brussels, the one that compared rotor performance with cascade performance, you made your comparison by taking essentially the total loss coefficient and subtracting from that an estimated shock loss. Then you added back in a mixing loss, which is what you were just talking about a minute or two ago. And then you found that you were able to correlate a typical loss parameter like loss coefficient times cosine β or sine β or whichever. You were able to correlate a loss parameter against a similar loss parameter as measured in cascade and that was a real good thing, I thought. I can't remember just exactly where your rotor measurements were made. These were not made right at the tip? These were made some distance in?

(Mikolajczak) We compared the rotor and cascade performance on the basis of total losses. The rotor losses were as measured; a mixing correction was added to the cascade measurements only, since these were taken very close to the cascade trailing edge. For interest only, we examined the simple NASA model, where you subtract the shock loss and correlate the viscous losses against diffusion factor. The data supported the NASA correlation. I don't know of anything basic that would substantiate why this correlation, derived from subsonic data, should hold for supersonic flow. It is a rule of thumb which works.

(Serovy) I am still trying to get Jim's question related to your answer. How far from the tip . . .

(Mikolajczak) We took our measurements at about 15% of the span from the tip. From the examination of the spanwise losses we felt we were outside of the wall loss region.

(Olson) You have also to raise the question as to the importance of radial flow shifts. What does the flow in the meridional plane do as it goes through the disc? For conditions where the streamline is not a cylindrical surface, the direct cascade analogy breaks down. The question is how far can you go in applying cascade data to rotor design systems for this case. I think that the data that Alex presented in his Brussels paper was for a case where the streamline was nearly parallel to the axis.

(Smith) There's another kind of problem that arises with relatively low aspect ratio rotor blades and a relatively low hub-tip radius ratio. Such blades have quite a bit of twist; and if you consider how a shock seen in the blade-to-blade pictures at different radii must be continuous, you find that you have a shock obliquity that shows up in the meridional plane. It seems to me that this is difficult to account for in a two-dimensional cascade also. We are going in this direction as rotor Mach numbers go up - up over two - of interest to NASA now - with radius ratios of around 0.5; and I think the blades that will do this are probably going to have aspect ratios less than two, maybe even considerably less than two. This other kind of obliquity is then going to be significant.

(Bullock) It certainly is going to be difficult to duplicate that kind of shock in a two-dimensional cascade.

(Mikolajczak) There is another possible problem in the transonic region. The work of McCune considers the Mach-one region of the blade. His analysis indicates that when there is a spanwise circulation gradient on the blade, the flow is very strongly three-dimensional in the sonic region. In extreme situations, his analysis would indicate negative camber where the normal design procedure requires positive camber. I wonder whether anybody has run a test in which there were spanwise circulation gradients in the transonic region, and whether any surprising results were observed.

(McBride) Before we complete our summation, I think it might be interesting to take a quick look at the effect of changes in streamline radius and axial velocity in an annular cascade. If we consider that, as in wing theory, the aerodynamic loading is represented by the chordwise local incremental or loading velocity; and assuming incompressible flow for simplicity, we get from moment of momentum:

$$\pm \frac{u'}{u_o} = \frac{1}{2\sigma} ctg\ \beta\ \cos\ \beta_o \left[\frac{2}{\varphi_o} \frac{d\nu}{d\xi} - \tau \xi^n \left\{ \frac{n tg\beta}{\xi} + \frac{dtg\beta}{d\xi} + \frac{tg\beta}{\nu} \frac{d\nu}{d\xi} \right\} - \frac{dtg\beta}{d\xi} + \frac{tg\beta}{\nu} \frac{d\nu}{d\xi} \right]$$

with notation:

subscript o denotes initial condition to cascade.

u' = local loading velocity related to pressure coefficient $\zeta \equiv \dfrac{(P_o - p)}{\frac{\rho}{2} v_o^2}$

$\zeta = 4 \dfrac{u'}{u_o} \cdot \dfrac{\bar{u}}{u_o}$

$\bar{u}$ = local reference velocity.

σ = cascade solidity.

φ_o = $C_a/\omega r_o$, inlet flow coefficient to rotor.

ν = r/r_o, dimensionless radius of streamline.

ξ = ℓ/c', dimensionless curved path length.

c' = curved chord length.

β = flow angle from axial.

C_a = axial velocity component.

ΔC_a = change in axial velocity across the cascade.

for convenience we define the axial velocity distribution through the cascade as:

$$Ca \equiv Ca_o + \Delta Ca \; (\xi)^n$$
$$= Ca_o \; \{1 + \tau \; \xi\}$$

where τ defines the magnitude of the axial velocity change (negative values denote deceleration) and the exponent n the distribution.

Since the local flow direction β is established principally by the cascade camber, it is obvious that a two-dimensional plane cascade cannot precisely model the load distribution of the actual machine with radial flow shifts unless either or both the camber distribution and axial diffusion are adjusted to compensate. It is interesting to note that the term $\frac{2}{\varphi_o} \frac{d\nu}{d\xi}$ vanishes for a stator so there is a difference between rotor and stator cascade loading unless $^{d\nu}/d\xi = 0$; i.e., no radial streamline shift.

(Lakshminarayana) Is u' your perturbation velocity?

(McBride) Yes, this is the local surface velocity perturbation, and it is related directly to the NASA pressure coefficient $\zeta = 4 \; u'/u_o \; \bar{u}/u_o$, where u_o is the initial velocity and $\bar{u}$ is the local remote wind speed. If I have no axial velocity change, the entire second term within the brackets goes out.

(Lakshminarayana) You should be differentiating the Cm_o which also changes through the cascade.

(McBride) The Cm_o by definition is the inlet velocity and is thus constant.

(Bullock) This comment pertains to the effects of turbulence. If we look at a first stage compressor rotor without inlet guide vanes, the turbulence effects should not be devastating. The following stator and second rotor should experience much higher turbulence, and the work of Schlicting and Das has shown that high free-stream turbulence improves the ability of a blade row to negotiate adverse pressure gradients. Their work also indicates that the heat transfer to turbines requires careful consideration of the level of turbulence.

(Smith) I would like to make a comment related to that. We found that when you vary the axial gap between adjacent blade rows, say make the gap very small so that the unsteadiness must be much greater, then the performance certainly doesn't get worse; it, in fact, gets remarkably better, so that this kind of unsteadiness which shouldn't be called turbulence isn't necessarily devastating to the performance.

(Olson) I think he was probably talking about a situation where the turbulence level can affect the transition point. There you might see important changes. On the other hand you have these cross changes in velocity which are called unsteadiness. It has been shown that an oscillating airfoil can go to higher angles of attack than an airfoil in steady flow. You have to distinguish between large-scale fluctuations and small-scale turbulence as they might affect the boundary layers.

(Bullock) Let us recall an experiment at the NACA about 20 years ago. A conventional turbine was tested, and its efficiency was measured. After considerable effort, the investigators learned how to put a set of nozzles far enough upstream so that an identical radial distribution of flow ahead of the rotor was achieved with no tangential gradients - such as wakes or secondary flow vortices. Much to the consternation of many people, the "wakeless" flow turbine efficiency was about three points lower than its conventional sister. This is the sort of phenomenon that cascades hide. Professor Vavra hit the nail on the head. When one understands the limitations of two-dimensional cascade data, they become an invaluable guide. One must be careful how he uses them, however. If you don't have a one-to-one comparison between two-dimensional data and turbomachine performance, you had better be prepared for some surprises.

(Olson) I don't think there is any real intensive effort to try to simulate turbulence levels in cascades. It seems to me that you have to distinguish between the small-scale turbulence, the size of which affect the boundary layer structure, and the large-scale turbulence, which maybe doesn't affect the structure.

(Papailiou) One should add that small-scale turbulence increases the overall losses. You make calculations with distributed turbulence, small-scale turbulence, and with the same velocity distribution and you see that the losses are increasing. However, most important is the effect of all kinds of turbulence on transition. Calculations indicate that for low levels of turbulence you get laminar separation bubbles; and in this case, because of the laminar separation bubbles, you get turbulent separation in the region of the trailing edge, which increases the losses. If you eliminate the laminar separation bubble by increasing either the macro-turbulence or the micro-turbulence, the turbulent separation can be eliminated. That may explain some of the kinds of differences that you get between cascade tests and compressor tests, where the turbulence levels are rather high.

(Olson) What kinds of calculations are these?

(Papailiou) Any kind of reasonably accurate, two-dimensional, laminar-turbulent calculation will do.

(Olson) What you are talking about is a transitional boundary layer method capable of predicting the effect of turbulence on transition or, in other words, whether you get a long bubble, laminar bubble, or a short bubble. If you say that this calculation is possible then I don't . . .

(Papailiou) No, what I mean is that if you have a laminar separation bubble that you can visualize, then you can calculate the laminar boundary layer and you can accept transition where it happens actually. Then you calculate from there on the turbulent boundary layer and you find out that it separates and you can make actual tests and find that the boundary layer is separating; while if you increase, as it happens in the actual compressor, either the macro- or the micro-turbulence, then you eliminate the laminar separation bubble and at the same time the turbulent separation. That should explain partly the difference in performance between cascades and rotors. I would like also to mention that there have been tests made on cascades using different levels of turbulence, and there has been simulation of turbulence by rods of the order of 5%, and what happens is that the main influence of turbulence is on the transition and not on the instability point. That means that the laminar boundary layer remains laminar as long as it is stable. That may help to design blades being sure that the laminar part remains laminar.

(Olson) What is missing in some of those experiments is an accurate measure of the turbulence spectra. What are the frequencies which are really affecting transition? While you might not find an effect in some particular turbulence-generating scheme or you did find that effect; is that effect then going to be present in the actual machine? In order to answer that question you have to determine what scales of turbulence are affecting transition and this business we were talking about before in trying to define the difference between small-scale turbulence, which affects transition, and the large-scale turbulence, which is an unsteady boundary layer, rather than say that it affects transition.

(Herring) I think you will find that the turbulence frequency bands in the wakes are wide enough so there is always a frequency around that will trigger transition. I don't think you have to worry about exactly what frequencies are present. Just a little way down from the trailing edge of the blade, you've got everything.

(Olson) In addition to the frequency it may also be that the amplitude is important. What I was suggesting is only that in experiments that are done to determine the effect of turbulence level, these experiments should have with them a complete mapping of the turbulence spectrum that was imposed on that blade, so that you would be able to draw general conclusions from it. I don't think we have that kind of data that we can . . .

(Herring) I don't think you do, but I am questioning whether that is necessary. I think you have any small disturbance you want.

(Oates) Isn't there a question also -- again getting back to the free-stream turbulence versus the shed turbulence that we were discussing -- that it would seem that the shed turbulence coming off the blade would obviously have frequencies of concern to the following blade because of the similar generating mechanism scales. But if we are going to put in free-stream turbulence upstream somewhere, I think we could put in, for example, r.m.s. disturbances that measure the same as one another at a different frequency that could have a pronouncedly different effect, because the generating mechanism would be quite different in the free stream.

(Herring) What do you mean by free stream?

(Oates) I envision this experiment that we were discussing about someone putting a turbulence-generating mechanism upstream of the cascade and studying the performance of the cascade as a function of this turbulence-generating mechanism. And I'm questioning whether we have to worry about the turbulence upstream so much, because almost anything will be available in a practical machine. I assume this is going to be of predominant importance in trying to obtain Reynolds-number effects for flying at high altitudes, where we would be getting serious changes in transition location. This becomes quite an important effect. Then if we were trying to simulate high-altitude turbulence scales, such turbulence would be different from that we would get in a laboratory. I think that you can have a power spectral density that has energies that appear to be at low frequency to a given phenomena when you are investigating it. If it is at low frequency, then we don't want to confuse it with ordinary turbulence. Or it could be considered turbulence on another scale. It could be atmospheric turbulence that people referred to.

(Unidentified) There is some older data, but there has been some recent work done at LTV in Dallas by Wells and some other people who have extended the previous NACA data down to much lower turbulence levels. They found a significant effect both on the onset of turbulence and on the distances required for the boundary layer to become fully turbulent as a function of free-stream turbulence level. This was reported in the AIAA journal a couple of years ago, five years ago I think.

(Olson) I remember that data in the Princeton series.

(Unidentified) Yes, that has been significantly extended to lower turbulence levels too by these more recent investigations.

(Herring) We are talking about very high turbulence level though, I mean compared to the data that . . .

(Unidentified) The point is that they are a function of turbulence level.

(Herring) Well, I am not so very sure that it is a strong function at the very high turbulence levels that we are talking about.

(Unidentified) What kinds of levels are you talking about? 5%?

(Smith) It is those kinds of levels when you are talking about wake passing.

(Unidentified) Well, there the point becomes even more germaine what the spectral content of the turbulence is, because the 5% level is a very gross average measurement.

(Smith) This is certainly not isotropic turbulence. And furthermore, every time we go from a rotor to a stator we change our frame of reference and, in a sense, pick up a uniform vector, the blade speed vector, that tends to suppress turbulence on a percentage basis. Consider the case of the absolute flow coming axially into a rotor. Now, the absolute flow may have 2% turbulence; but if the relative inlet flow angle is 60°, a typical value, the turbulence level relative to the rotor would be 1%.

(Olson) Maybe we should move on to radial-equilibrium calculations.

SUPERSONIC FLOW IN CASCADES

Discussion Leader: Mr. R. E. Olson

Presentation: Radial Equilibrium Across a Normal Shock in an Axial Rotor

by

Mr. P. Schwaar

RADIAL EQUILIBRIUM ACROSS A NORMAL SHOCK IN AN AXIAL ROTOR

by P. Schwaar

The problem I suggested to discuss here is such a simple one that I am wondering now whether it fits properly into the frame of the discussions of this workshop. It is concerned with the simple radial-equilibrium condition for the flow across a normal shock wave in the rotor blading. The problem is not new altogether. In 1954 I treated the shock-in-stator case and showed that, in the assumption of axisymmetric flow conditions, the shock surface must be a radial helicoidal surface in order to be perpendicular to the incoming flow direction everywhere. This determines the flow angle over the stator channel height, which happens to be that of a so-called free-vortex flow, i.e., $\tan \beta = Cr$. The radial-equilibrium condition then determines the radial distribution of the relative Mach number of the incoming flow, which results in a radially non-constant absolute stagnation enthalpy. In 1956, I derived the corresponding equations for the shock-in-rotor case. In 1959, Hammitt and Bogdonoff treated the problem in essentially the same manner in an ASME paper. They extended the treatment to oblique shocks, and also relaxed the condition of shock perpendicularity everywhere by departing from the free-vortex flow angle condition. Recently we have been interested in the design of a centrifugal compressor with supersonic inducer section. This prompted me to review the subject and to solve the equations I had derived in 1956.

Figure 1 shows the velocity triangle at rotor entrance (in front of the shock wave) and lists the fundamental equations which describe the situation. Equation (1) is the simplified radial-equilibrium condition, written in terms of the Mach number of the tangential component V_u of the absolute velocity V. Equation (2) relates the static pressures $\hat{p}$ and p after and in front of the shock;
Equation (3), the corresponding relative Mach numbers $\hat{M}_w$ and M_w; and Equation (4), the static temperatures $\hat{t}$ and t. Logarithmic differentiation of Equation (2), introduction of Equation (1) on both sides of the shock, and replacing $\hat{M}_w^2$, t and $\hat{M}_u$ by Equations (3) and (4) yields differential Equation (5), for which we consider two cases, as explained on Figure 2.

In case 1, the relative inlet flow angle β is not specified and is generally defined by Equation (6a). In case 2, β is specified according to Equation (6b), in which, for example, n = 0 calls for β = const. and n = 1, for free-vortex radial evolution. Equation (5) now can be formally integrated and put in form of integral Equation (7), where F(r) is defined by Equation (8a) in case 1 and by Equation (8b) in case 2.

Additional relations shown on Figure 3 are needed to determine the problem. The isentropic relation (9) combined with the radial-equilibrium condition (1) yields Eq. (10) for the ratio of the stagnation and the static temperatures in front of the shock. Equation (11) is the energy relation. Equation (12) expresses the velocity triangle relation. Finally, if we assume that the stagnation state in front of the shock has been generated by a polytropic compression through preceeding stages, we can relate the stagnation pressure P in front of the shock to the initial stagnation pressure P_o through Eq. (13) where η_p is the polytropic efficiency of the compression process, which can be a prescribed function of r.

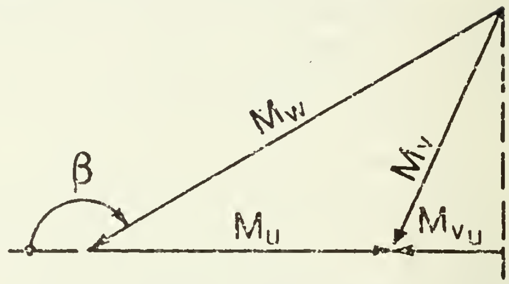
We have now five equations, namely (7), (10), (11), (12), and (13). In case 1, we deal with six unknowns, and one quantity can be freely selected. Case 2 has only five unknowns since M_{vu} is related to β by Eq. (14), and the problem is thus completely determined. The solution is arrived at in both cases by alternate iterations of Eqs. (7) and (10), passing from one to the other by means of Eqs. (11), (12), (13), and (14). I do not go here into the details of this calculation procedure.

Two illustrative examples have been worked out and are reproduced on Figure 4. For the case of an unspecified β, the absolute stagnation state (P,T) in front of the shock has been assumed uniform. For the second case, β has been assumed constant over the radius. The velocity triangles shown on Figure 4 correspond to a hub/tip ratio of .733, and show that in both cases the relative Mach number M_w in front of the shock varies only slightly along the radius. This, in the case of a negative prerotation of the absolute inlet velocity, can only be induced by a sharp decrease of the tangential velocity component over the blade height. In fact, this decrease is faster than for free-vortex flow conditions according to $V_u r$ = const. This is evidenced by comparing the tip V_u-components with those corresponding to free-vortex with equivalent hub V_u-components, which are indicated by the vertical dash on the blade tip speed segment. Such flow conditions, however, can be shown to be unstable in the sense that if a fluid particle is artificially displaced from its radial location, it will not return to its original location but will continue its radial displacement in the same sense. This instability is more pronounced for the case of an unspecified β than for β= const. In order to provide some insight into this problem, we made a circular cascade test with a setup consisting of two consecutive stationary blade rows. The first cascade imparts to the air a tangential velocity distribution similar to that imparted by an axial rotor preceeding the supersonic rotor. The second cascade decelerates the flow and generates the tangential velocity component required for radial equilibrium through a normal rotor shock wave according to the more critical case 1. The results show that the calculated flow conditions are closely realized at exit of the second cascade (instrumentation plane 2.0) and that no deterioration of the flow configuration takes place downstream of that cascade (plane 3.0).

As I mentioned before, this is a very simple case of radial-equilibrium flow calculation. It does not provide an answer to the problem mentioned by Mr. Bullock, namely that of placing a shock wave in a low hub-tip ratio rotor with subsonic hub and supersonic tip flow conditions.

FIGURE 1. SIMPLIFIED RADIAL EQUILIBRIUM ACROSS NORMAL SHOCK IN ROTOR

1. Radial Equilibrium :



$$(1) \qquad \frac{1}{p}\frac{dp}{dr} = \gamma \frac{M_{v_u}^2}{r}$$

$$M_{v_u} = M_w \cos\beta + M_u$$

2. Normal Shock Relations :

$$(2) \qquad \frac{\hat{p}}{p} = 1 + \frac{2\gamma}{\gamma + 1}(M_w^2 - 1)$$

$$(3) \qquad \hat{M}_w^2 = \frac{2 + (\gamma - 1) M_w^2}{2\gamma M_w^2 - (\gamma - 1)}$$

$$(4) \qquad \frac{\hat{t}}{t} = \frac{\left[2 + (\gamma - 1) M_w^2\right]\left[2\gamma M_w^2 - (\gamma - 1)\right]}{(\gamma + 1)^2 M_w^2}$$

$$\left(\hat{M}_u = M_u \sqrt{\frac{t}{\hat{t}}}\ \right)$$

3. Differential Equation :

$$(5) \qquad \frac{\frac{dM_w^2}{dr}}{M_w^2 - 1} + (1+\gamma M_w^2)\frac{\cos^2\beta}{r} + 2\gamma M_w M_u \frac{\cos\beta}{r} + \frac{\gamma - 1}{2}\frac{M_u^2}{r} \quad \frac{1+\gamma M_w^2}{1 + \frac{\gamma - 1}{2} M_w^2} =$$

FIGURE 2. SOLUTION OF RADIAL EQUILIBRIUM EQUATION

CASE 1	CASE 2
β unspecified:	β specified:
6a) $\cos\beta = \dfrac{W_u}{W} = \dfrac{M_{v_u} - M_u}{M_w}$	6b) $\mathrm{tg}\beta = C.r^n$

Integral Form:

$$(7) \qquad M_w^2\,(r) = 1 + (M_{w_i}^2 - 1)\ \exp\left[\int_{r_i}^{r} F(r)dr\right]$$

Case 1:

$$8a) \qquad F(r) = \frac{M_u^2}{M_w^2 \cdot r} \cdot \frac{M_w^2 - 1}{1 + \frac{\gamma - 1}{2} M_w^2} + \frac{2\,M_{v_u}}{M_u} - \frac{M_{v_u}^2}{M_u^2}\ (1 + \gamma\,M_w^2)$$

Case 2:

$$8b) \qquad F(r) = \frac{2\gamma\,M_w M_u}{r(1 + C^2 r^{2n})^{1/2}} - \frac{1 + \gamma\,M_w^2}{r(1 + C^2 r^{2n})} \cdot \frac{\gamma - 1}{2} \cdot \frac{M_u^2}{r} \cdot \frac{1 + \gamma\,M_w^2}{1 + \frac{\gamma - 1}{2} M_w^2}$$

FIGURE 3. ADDITIONAL RELATIONS

(9) $$\frac{P}{p} = \left[\frac{T}{t}\right]^{\frac{\gamma}{\gamma-1}} \text{ combined with } \frac{1}{p}\cdot\frac{dp}{dr} = \gamma\,\frac{M_{v_u}^2}{r}$$

(10) $$\left[\frac{T}{t}\right]_r = \left[\frac{T}{t}\right]_i \cdot \left[\frac{P}{P_i}\right]^{\frac{\gamma-1}{\gamma}} \exp\left[-(\gamma-1)\int_{r_i}^{r}\frac{M_{v_u}^2}{r}\,dr\right]$$

(11) $$\frac{T}{t} = 1 + \frac{\gamma-1}{2} M_v^2$$

(12) $$M_w^2 = M_v^2 + M_u^2 - 2\,M_u\,M_{v_u}$$

(13) $$P = P_o\left[\frac{T}{T_o}\right]^{\eta_p\frac{\gamma}{\gamma-1}}$$

Case 1: 6 Unknowns: M_w, M_v, M_{v_u}, T, t, P
Select one quantity, f. ex. P(r)

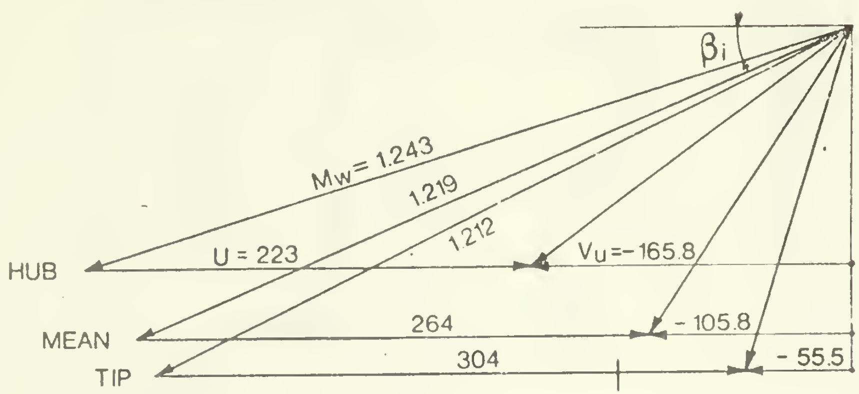
Case 2: $$M_{v_u} = M_w\cos\beta + M_u = M_u - \frac{M_w}{(1 + c^2 r^{2n})^{1/2}} \qquad (14)$$

5 Unknowns: M_w, M_v, T, t, P

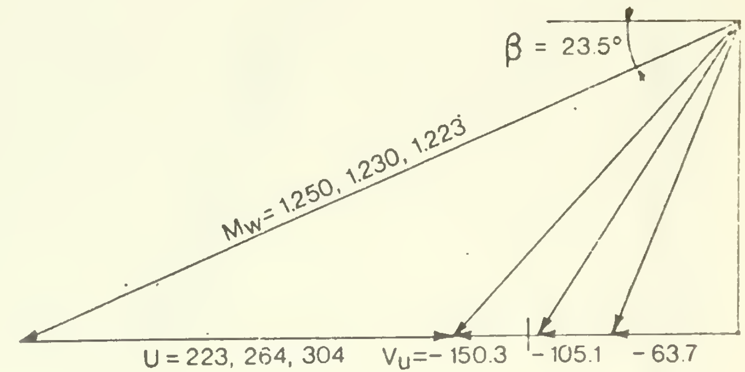
FIGURE 4. ILLUSTRATIVE EXAMPLES

CASE 1: $\beta(r)$ Unspecified, Uniform Absolute Entrance Stagnation State

$\nu = 0.733$

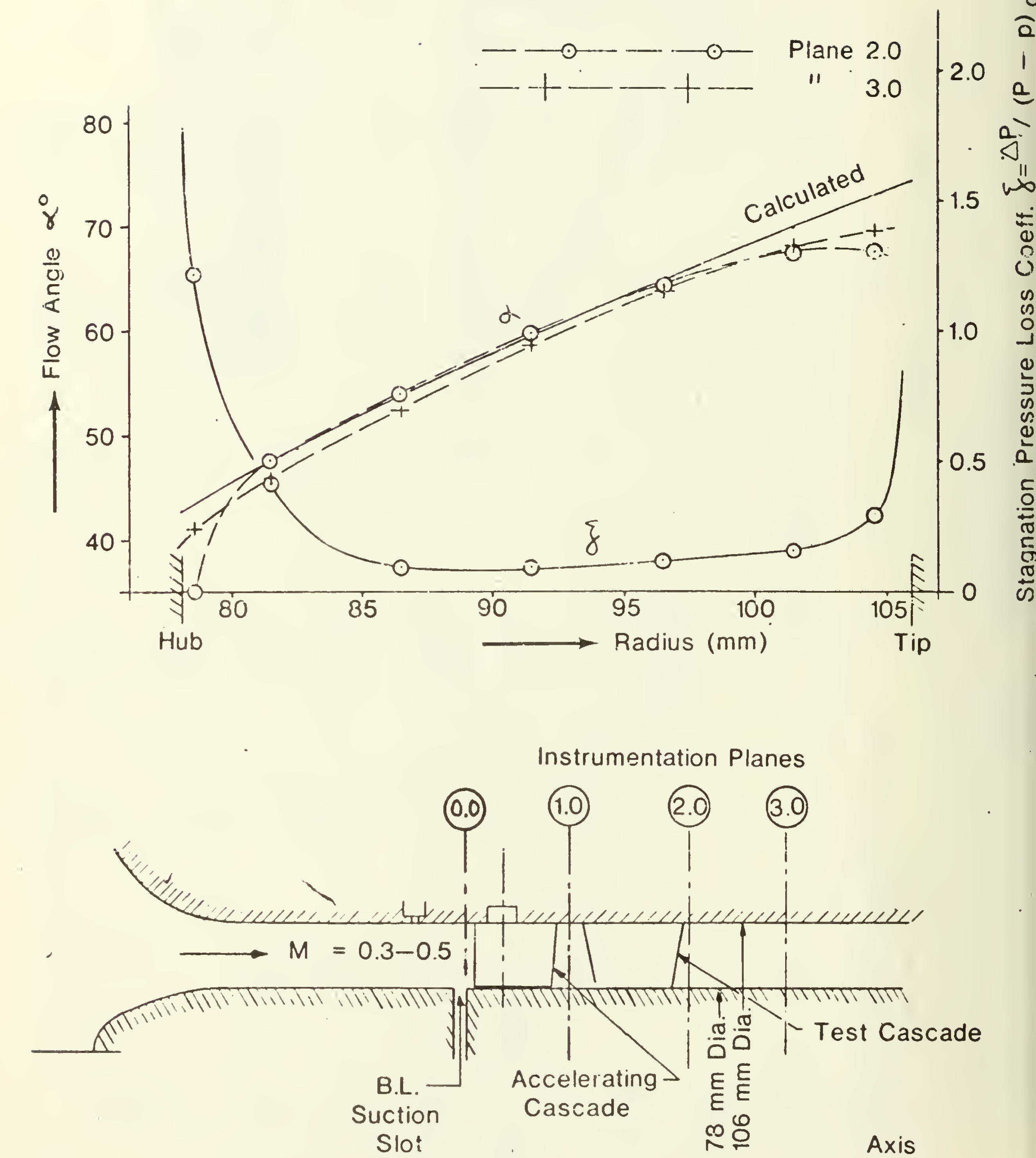


CASE 2: $\beta(r)$ = Const. = 23.5°



Absolute Entrance Stagnation State	HUB	MEAN	TIP
P/P_i	1	0.962	0.932
T/T_i	1	0.989	0.980

FIGURE 5. PREROTATION FLOW CASCADE TEST



ACTUAL THREE-DIMENSIONAL FLOW PATTERNS

Discussion Leader: Dr. J. R. Fagan

THREE-DIMENSIONAL, INVISCID FLOW ANALYSIS IN TURBOMACHINERY*

by

John R. Fagan

* Editor's Comment: This is the paper on which Dr. Fagan based his presentation.

THREE-DIMENSIONAL, INVISCID FLOW ANALYSIS IN TURBOMACHINERY

Introduction:

Evaluation of current turbomachinery designs shows that steady, three-dimensional, inviscid, rotational and compressible flow play a significant and fundamental part in the interpretation of observed performance. As an example of the strong three-dimensional aspects of flow in turbomachines, consider Figure 1 which shows the stage, absolute, total pressure contours developed by a tandem rotor obtained by probe measurements downstream of the stator. In spite of this, the complexity of this problem has precluded the development of analytical methods for predicting this class of flows. Mathematically this difficulty stems principally from the necessity of solving two very complex momentum equations simultaneously. Three basic approaches to this problem have been suggested in the literature. The most general approach is the particle-in-cell (PIC)[(1)*] method. This method is handicapped by very large computer memory and storage time requirements as are all direct 3-D differencing techniques. The recently published work by Stuart and Hetherington[(2)] uses iterative two-dimensional calculations based upon a streamline curvature formulation. This approach has successfully obtain

---

[*] See references at the end of this paper.

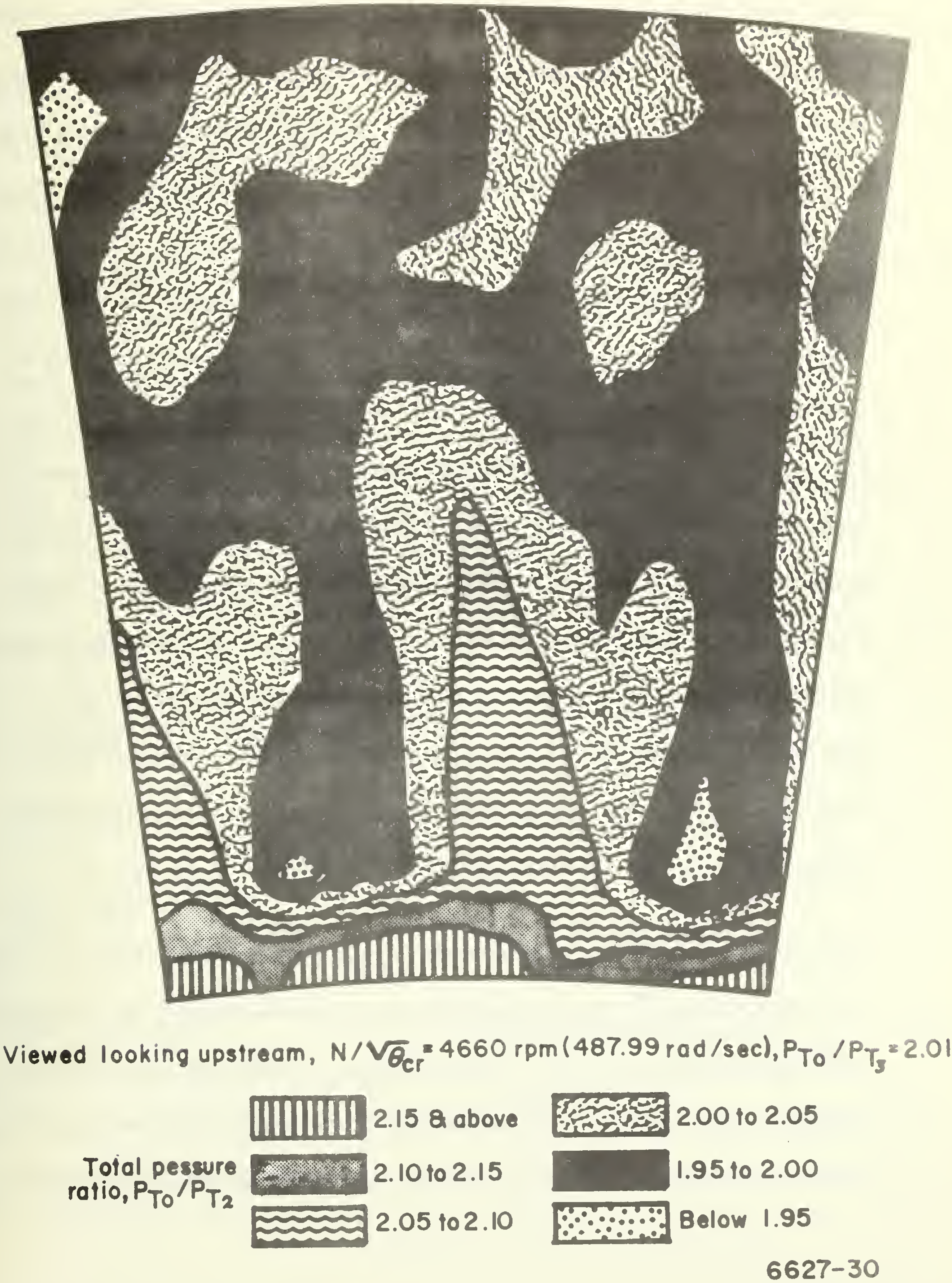


Figure 1. Turbine stage total pressure ratio contours for low solidity tandem rotor blade turbine.

convergence solutions for the full three-dimensional flow field and gave good agreement with experimental results. The work at Detroit Diesel Allison Division also is predicated upon an iterative solution of mathematically two-dimensional problems. Unlike the work of Stuart and Hetherington, it is based fundamentally on a stream function solution for the flow on stream surface located arbitrarily in space. The essential element of this calculational procedure is a fast, stable and efficient solution technique for Poisson-type equations. A procedure termed "accelerated iteration by lines" has been adopted for use. This algorithm has proven stable as well as significantly reducing computer memory requirements and reducing operating time by approximately 30% for typical flow analyses when compared with matrix-reduction methods. One feature of this stream-function formulation on a arbitrary stream surface which is proving particularly valuable is the capability to evaluate the effects of stream surface distortion (inviscid secondary flow effects) in either the meridional or blade-to-blade plane without establishing the entire 3-D flow field.

The essential difficulty established in this work as well as that of Stuart and Hetherington[2] is communicating between one set of two-dimensional calculations and the other. This so-called "communication problem" has introduced new stability and convergence difficulties into the calculation associated specifically with the three-dimensional aspects of the problem

Three approaches to this "communication problem" have been formulated and are under investigation.

Governing Equations:

Steady, strictly adiabatic, inviscid flow of an ideal gas is frequently taken as the basis for fluid mechanics analysis of flow in turbomachines. Accepting this basis, the continuity equation, the three momentum equations, the energy equation, the equation of state, and the defining equation for total stagnation enthalpy are the governing equations. These seven equations then form a closed system for the three velocity components, $(v_x, v_y \,\&\, v_z)$, the density (ρ), the entropy (S), the temperature (T) and the total stagnation enthalpy (H). Thus,

$$\nabla \cdot \rho \vec{v} = 0 \qquad \text{continuity} \qquad (1)$$

$$-\vec{v} \times (\nabla \times \vec{v}) = \nabla H - T \nabla S \qquad \text{momentum} \qquad (2)$$

$$(\vec{v} \cdot \nabla) H = 0 \qquad \text{energy} \qquad (3)$$

$$\frac{\rho}{\rho_0} = \left(\frac{H - v^2/2}{h_0}\right)^{1/\gamma - 1} e^{-\frac{S - S_0}{R}} \qquad \text{equation of state} \qquad (4)$$

$$H = C_p T + \frac{1}{2} v^2 \qquad \text{total stagnation enthalpy} \qquad (5)$$

The primary mathematic difficulties in solving this set of equations arise from the simultaneous solution of the three complex momentum equations.

This difficulty can be alleviated by forming the vector dot product of $\vec{v}$ with the momentum equation and subtracting the energy equation to show

$$(\vec{v}\cdot\nabla)S = 0$$

This equation, which states the entropy is constant along a streamline, can be used to replace one of the momentum equations. The modified set which contains (6) in place of one of the momentum equations (2) still requires the simultaneous solution of two momentum equations. As a matter of expediency, one of the momentum equations is usually eliminated at this point by assuming two-dimensional flow (i. e., one of the velocity components is zero). C. H. Wu[3] pointed out that three-dimensional flow fields can be built up from mathematically two-dimensional calculations by applying the analysis on a predetermined but arbitrary stream surface with variable stream tube thickness. In this procedure the equation of the stream surface replaces the second of the three momentum equations. The stream surface geometry is systemmatically updated as the calculation progresses from one two-dimensional problem to the next.

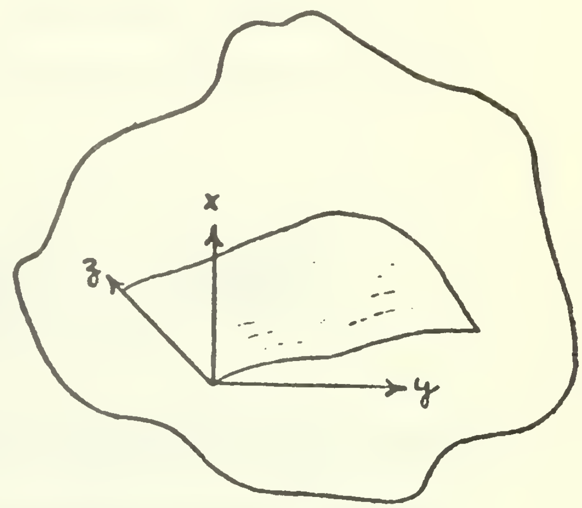
Solution on an Arbitrary Stream Surface:

The application of the governing equations to a predetermined stream surface is illustrated in Figure 2 for the continuity equation. Let

$$x = f(y, z)$$

Figure 2. Continuity on a Stream Surface

$$\frac{\partial \rho v_x}{\partial x} + \frac{\partial \rho v_y}{\partial y} + \frac{\partial \rho v_z}{\partial z} = 0 \qquad x = f(y, z)$$



$$\frac{\bar{\partial}}{\partial y} = \frac{\partial}{\partial y} + \frac{\partial}{\partial x}\frac{\partial x}{\partial y}$$

$$\frac{\bar{\partial}}{\partial z} = \frac{\partial}{\partial z} + \frac{\partial}{\partial x}\frac{\partial x}{\partial z}$$

$$\frac{\bar{\partial} \rho v_y}{\partial y} + \frac{\bar{\partial} \rho v_z}{\partial z} = -\rho\left(\frac{\partial v_x}{\partial x} - \frac{\partial v_y}{\partial x}\frac{\partial x}{\partial y} - \frac{\partial v_z}{\partial x}\frac{\partial x}{\partial z}\right)$$

$$\frac{\bar{\partial}(b\rho v_y)}{\partial y} + \frac{\bar{\partial}(b\rho v_z)}{\partial z} = 0$$

$$\frac{\partial \Psi}{\partial y} = b\rho v_z \qquad \frac{\partial \Psi}{\partial z} = -b\rho v_y$$

be the defining equation of the stream surface where f(y, z) is known. Introducing the special derivatives $\frac{\bar{\partial} q}{\partial y}$ and $\frac{\bar{\partial} q}{\partial z}$, which denote the rate of change of any quantity, q, on the stream surface with respect to y and z with the other held constant, and an integrating factor, b, which can be shown to be proportional to the stream tube thickness, into the continuity equation allows it to be written as

$$\frac{\bar{\partial}(b\rho v_y)}{\partial y} + \frac{\bar{\partial}(b\rho v_z)}{\partial z} = 0 \qquad (8$$

In this form the continuity equation is written entirely in terms of flow properties on the stream surface, and a stream function, ψ, can be defined as shown in Figure 2. Substituting these special derivatives and the stream function in the one remaining momentum equation produces equation (I) of Figure 3. Equations (II) and (III) of Figure 3 are the energy and entropy equations, respectively. On the stream surface the total stagnation enthalpy, H, and the entropy, S, are functions of the stream function, ψ, alone; and this functional relationship can be established from the inlet flow conditions. This remaining momentum equation (I) is a second-order, quasi-linear, partial differential equation in the stream function, ψ. The solution is initiated by obtaining a first estimate of the stream function for incompressible, irrotational flow (F=0). This

Figure 3. Stream Surface Theory

(I) $$\frac{\bar{\partial}^2 \Psi}{\partial y^2} + \frac{\bar{\partial}^2 \Psi}{\partial z^2} = F(H, S, \Psi)$$

(II) $$H = f_1(\Psi)$$

(III) $$S = f_2(\Psi)$$

Three scalar equations in three unknowns:

$$H, S, \Psi$$

supplemented by the auxiliary relations

$$F(H,S,\Psi) = \frac{b\rho}{\dot{m} v_z}\left[\frac{\bar{\partial} H}{\partial y} - T\frac{\bar{\partial} S}{\partial y} + \frac{n_y}{n_x}\frac{1}{\rho}\frac{\partial p}{\partial x} - v_x \frac{\bar{\partial} v_x}{\partial y}\right]$$

$$+ \frac{\bar{\partial} \Psi}{\partial y}\frac{\bar{\partial} \ln b\rho}{\partial y} + \frac{\bar{\partial} \Psi}{\partial z}\frac{\bar{\partial} \ln b\rho}{\partial z}.$$

$$\frac{\rho}{\rho_0} = \left(\frac{H - v^2/2}{h_0}\right)^{1/\gamma - 1} e^{-\frac{S - S_0}{R}}$$

$$\frac{\bar{\partial} \Psi}{\partial y} = b\rho v_z \qquad \frac{\bar{\partial} \Psi}{\partial z} = -b\rho v_y$$

initial Ψ value is then used in equations (II) and (III) and the auxiliary relations of Figure 3 to obtain an updated F distribution. This updated F distribution is then used in equation (I) to improve the Ψ estimate, and this iterative procedure is continued until a consistent set of F values are obtained. It should be noted that the differential surface elements ($\partial x/\partial y$)$_{z=\text{constant}}$ have been replaced by their equivalents in terms of unit normal surface vector components, n_y/n_x .

The essential element in this computation is a fast, stable and efficient solution technique for Poisson-type equations (equation I of Figure 3). A procedure termed "accelerated iteration by lines" has been adopted for use. As illustrated in Figure 4, the "accelerated-iteration-by-lines" technique is based upon introducing a relaxation factor, λ , into the central difference approximation to the Poisson equation and rearranging the terms to obtain an iteration equation. This iteration equation is so arranged as to require only data from the previous iteration and upstream columns to obtain the new Ψ estimates for any column. Thus, it is possible to march through the flow field from front to rear solving for each column successively. This iteration is continued until the maximum change in Ψ from one pass to the next is sufficiently small.

Figure 5 illustrates the procedure used to solve for each node within a given column. First, the stream function at any node, n+1, in column, m,

Figure 4. Accelerated Iteration By Lines

The general form of the simultaneous algebraic equations satisfying Poisson's equation at a grid point m,n can be written as

$$\Psi_{m+1,n} - (2-\lambda)\,\Psi_{m,n} + \Psi_{m-1,n} + \Psi_{m,n+1} - (2+\lambda)\,\Psi_{m,n} + \Psi_{m,n-1} = h^2 F_{m,n}$$

This can be written as an iteration equation where t is the iteration number. Thus,

$$\left(\Psi_{m,n+1} - (2+\lambda)\,\Psi_{m,n} + \Psi_{m,n-1}\right)^{t+1} = -\Psi^{t}_{m+1,n} + (2-\lambda)\,\Psi^{t}_{m,n} - \Psi^{t+1}_{m-1,n} + h^2 F_{m,n} = P^{t}_{m,n}$$

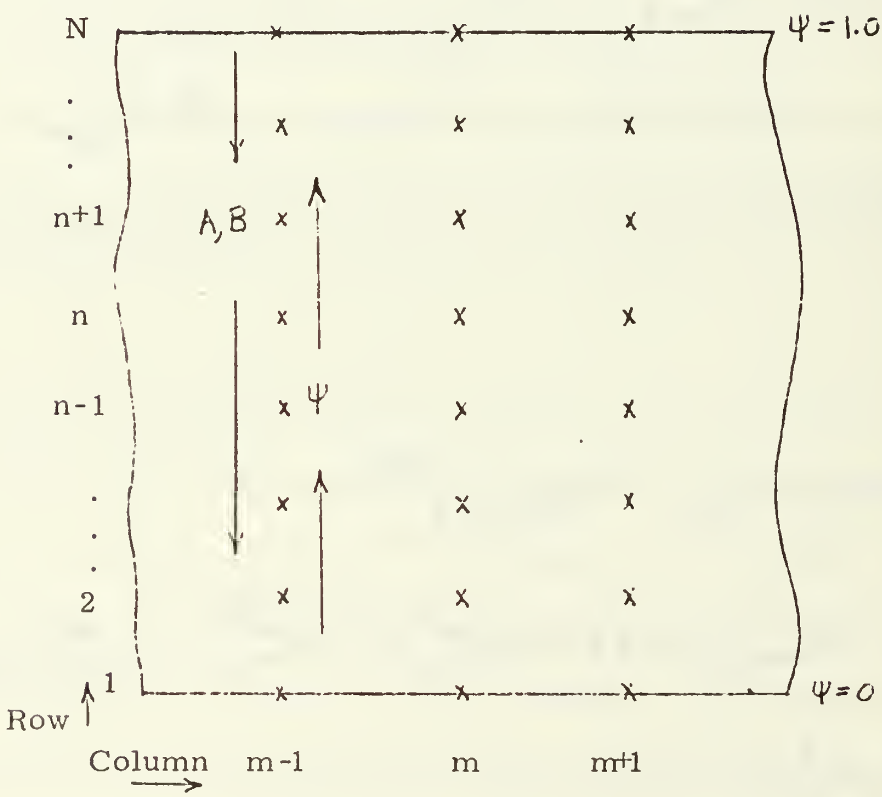
This makes it possible to march through the flow field from front to rear solving for a whole line at one time.

Figure 5. AIL Calculation Procedure

$$\frac{\partial^2 \psi}{\partial y^2} + \frac{\partial^2 \psi}{\partial z^2} = F(y, z)$$

$$a\,\Psi_{m,n-1} + b\,\Psi_{m,n} + c\,\Psi_{m,n-1} = P_{m,n}$$

$$\Psi_{m,n+1} \equiv A_{m,n} + B_{m,n}\,\Psi_{m,n}$$



$$\Psi_{m,N} = 1.0 = A_{m,N-1} + B_{m,N-1}\,\Psi_{m,N-1} \Rightarrow A_{m,N-1} = 1.0;\ B_{m,N-1} = 0$$

$$A_{m,n-1} = \frac{P_{m,n} - c\,A_{m,n}}{b + c\,B_{m,n}} \quad ; \quad B_{m,n-1} = \frac{a}{b + c\,B_{m,n}}$$

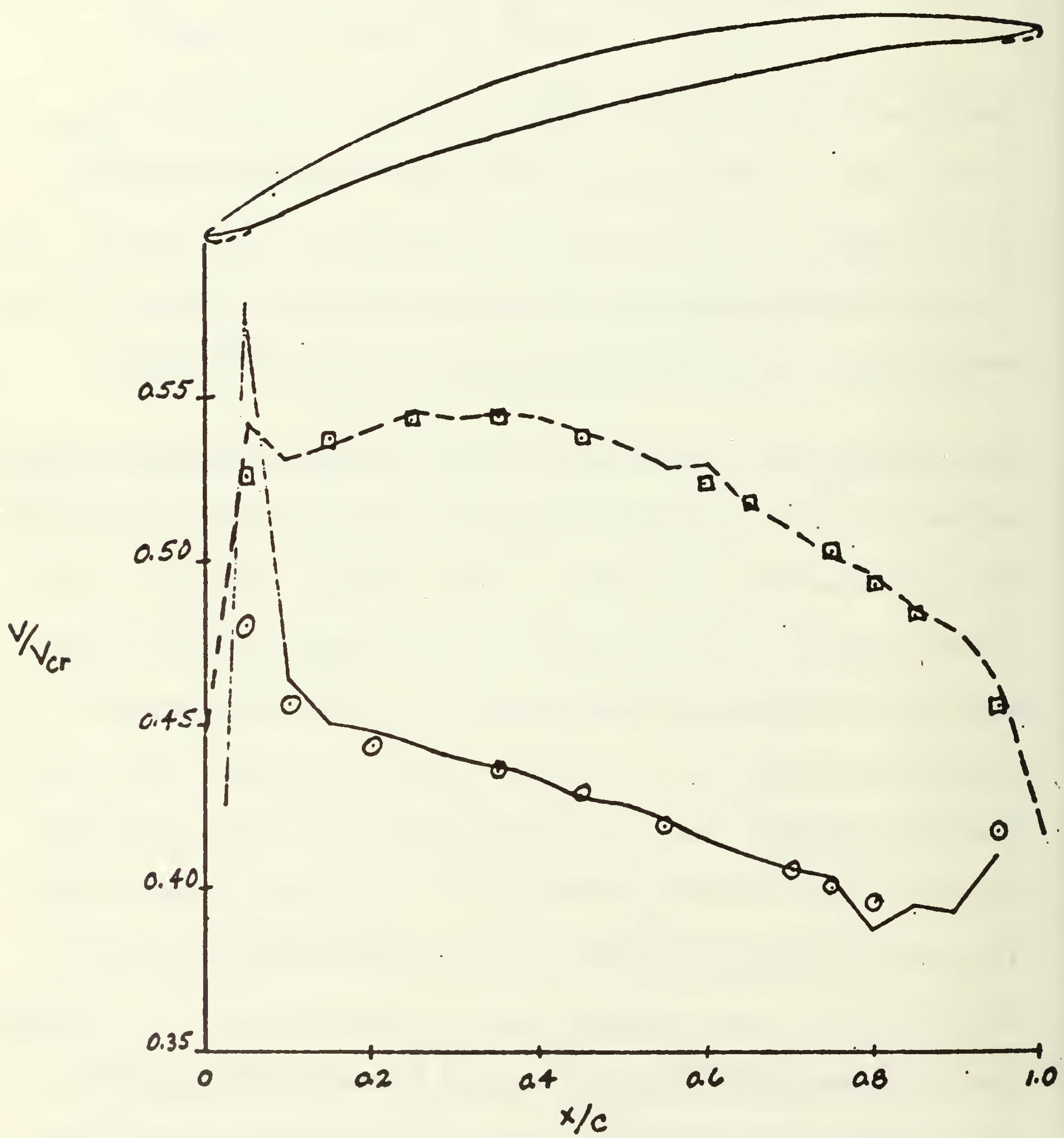
is assumed to be related to the value of Ψ at node, n, in that column by

$$\Psi_{m,n+1} = A_{m,n} + B_{m,n} \Psi_{m,n} \tag{9}$$

Noting the boundary condition, $\Psi_{m,N} = 1.0$, on the top surface, $A_{m,N-1}$ is set equal to 1.0 and $B_{m,N-1}$ is set to zero. As shown in the bottom line of Figure 5, the position-dependent coefficients, $A_{m,n}$ and $B_{m,n}$ can be obtained successively from the top to the bottom of the column. These coefficients are then used in equation (9) to obtain the Ψ values.

The calculation procedure described above for flow on an arbitrary stream surface has been compared to the blade-to-blade calculation of Katsanis[4] for a two-dimensional, circular-arc blade element. The results of this comparison are shown in Figure 6. Good agreement was obtained for the surface velocity distributions except near the leading edge, where the rather large mesh spacing used in the current calculations did not accurately reproduce the blade shape. Calculations were also made to determine the effect of stream surface warpage on the flow field. Figure 7 shows the pressure distributions obtained from a parametric investigation of stream surface warpage and transverse pressure gradients. The channel geometry used for this study represents a low-turning blade cascade with straight ducts in front and behind it. The exit flow direction was

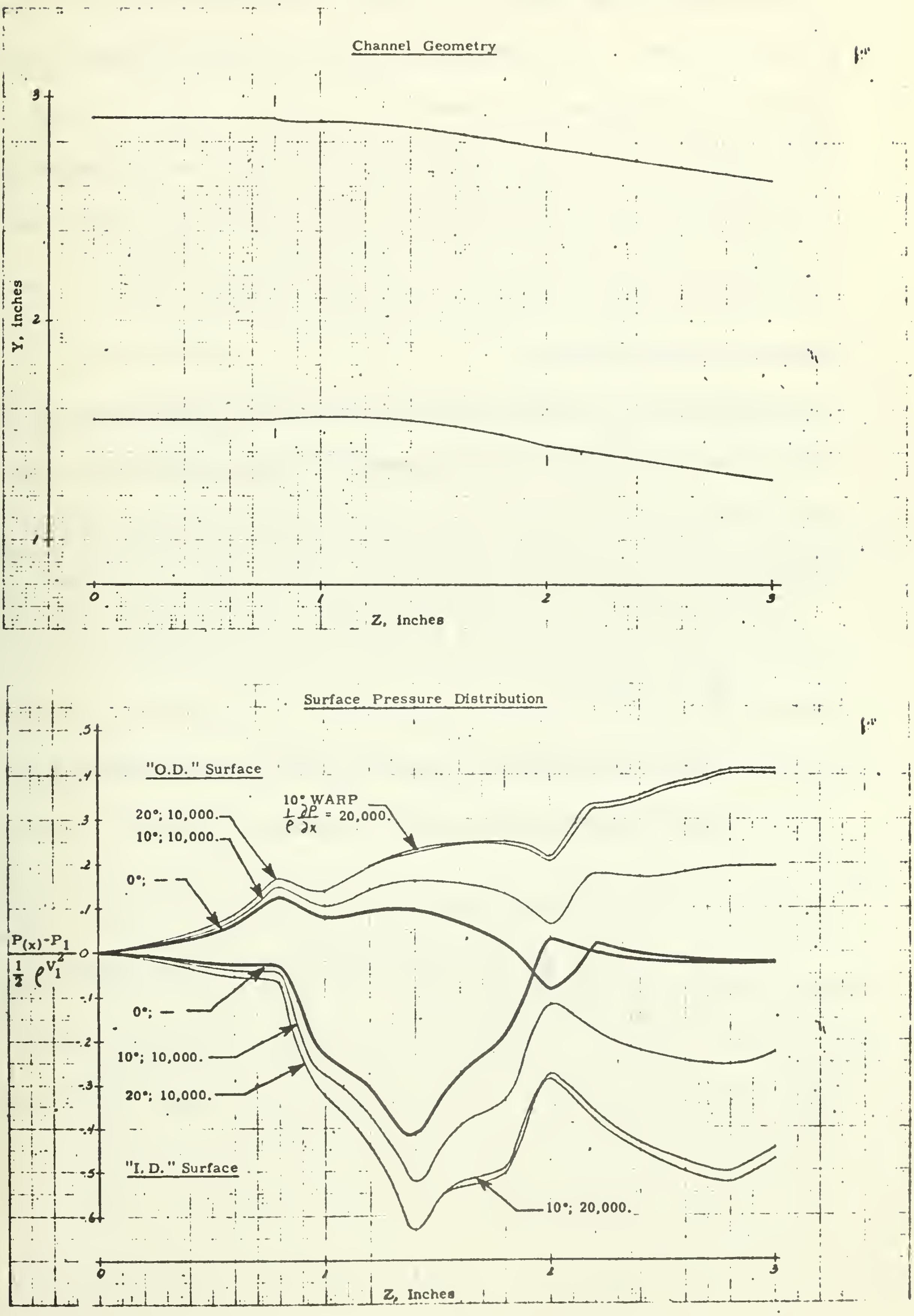
Figure 6. Comparison of AIL Program With The Work of Katsanis



⊡ ⊙ Katsanis Program Code (Ret. 4)

---- Current Work

Figure 7. Stream Surface Warpage Effects



set to establish the same static pressure on the top and bottom walls of the exit duct to simulate the periodicity of a blade cascade for two-dimensional flow. The periodicity is destroyed when stream surface warpage is allowed, and as a result the exit static pressures are no longer equal. The results of this study show that marked changes in the surface pressure distribution can occur when stream surface warpage occurs in conjunction with a transverse pressure gradient.

Three-Dimensional Flow:

The availability of a computational procedure applicable to flow on a general two-dimensional surface makes it possible to build up a full three-dimensional flow field from mathematically two-dimensional calculations. The iterative computation would proceed as follows:

1. Assume the geometry of an initial set of stream surfaces, ϕ = constant.

2. Solve the 2-D flow problem on these stream surfaces to obtain the streamlines, Ψ = constant.

3. Connect the streamlines of like value (i.e., Ψ = .2, .4, etc.) on the set of ϕ surfaces to define a new set of stream surfaces, Ψ = constant.

4. Solve the 2-D flow problem on these new surfaces which are essentially normal to the original Φ surfaces for the streamlines, Φ = constant.

5. Test the location of the new stream surfaces against those of the previous iteration and the axial velocity component throughout the flow field from the previous half iteration (i. e., from Ψ = constant). If these are sufficiently small the solution is converged, if not, repeat the 2-D calculation continually updating the stream surface geometry until convergence is obtained.

Whereas in principle the above procedure leads to definition of the complete 3-D flow field, in practice both stability and convergence problems prevent solution when initial estimates of the stream surface geometry deviate appreciably from the resultant solution. Experience at DDAD as well as the reported work of Stuart and Hetherington[2] have identified as critical the transfer of information from calculations on the Φ streamsurfaces to calculations on the Ψ surfaces. This "communication problem" exists predominantly because the transfer of stream surface geometry (or unit normals $\frac{n_y}{n_x}$ & $\frac{n_z}{n_x}$) and transverse pressure gradient ($\partial P/\partial x$) information is inadequate to force convergence. This is seen most clearly when plane surfaces are used as an initial guess. In this case the unit normals n_y & n_z are zero and no transverse pressure gradient information is transferred at all. This is seen clearly in the first equation of Figure 8. In any turning duct, the unit normals, n_y & n_z, will not be zero simultaneously. However, the axial momentum equation (z-momentum) is far less stable numerically than the momentum equation taken across the flow field. This results directly from the very small (or even zero) values of the cross-flow velocity, v_y. It should be remembered that it was possible to eliminate one of these equations because it was replaced by the entropy equation. In principle, both of them must be satisfied simultaneously as shown in the second equation of Figure 8. The simultaneous solution can be shown analyticall to mean that the assumed stream surface geometry exactly satisfied the transverse (or x) momentum equation.

The critical lack of information appears to result directly from the assumption that the calculation surfaces are stream surfaces. A strictly Eulerian iteration scheme is shown in Figure 9 which does not make this assumption. The primary change in the computation is in the calculation of the entropy, S, and total stagnation enthalpy, H, derivative. On a stream surface one of the stream functions, Ψ or ϕ, was constant. In the strictly Eulerian system H and S are functions of both Ψ and ϕ o the computation surface. The basic iteration procedure, shown in Figur 9, consecutively updates the stream functions on a 3-D array of grid nodes

Figure 8. The Communication Problem

$$\frac{\bar{\partial}^2\psi}{\partial y^2} + \frac{\bar{\partial}^2\psi}{\partial z^2} = \frac{\bar{\partial}\psi}{\partial y}\frac{\bar{\partial}\ln b\rho}{\partial y} + \frac{\bar{\partial}\psi}{\partial z}\frac{\bar{\partial}\ln b\rho}{\partial z}$$

$$+\frac{b\rho}{\dot{m}v_z}\left[\frac{\bar{\partial}H}{\partial y} - T\frac{\bar{\partial}S}{\partial y} + \frac{n_y}{n_x}\frac{1}{\rho}\frac{\partial p}{\partial x} - v_x\frac{\bar{\partial}v_x}{\partial y}\right] \quad \text{y-momentum}$$

or

$$+\frac{b\rho}{\dot{m}v_y}\left[-\frac{\bar{\partial}H}{\partial z} + T\frac{\bar{\partial}S}{\partial z} - \frac{n_z}{n_x}\frac{1}{\rho}\frac{\partial p}{\partial x} + v_x\frac{\bar{\partial}v_x}{\partial z}\right] \quad \text{z-momentum}$$

Thus,

$$\frac{\bar{\partial}^2\psi}{\partial y^2} + \frac{\bar{\partial}^2\psi}{\partial z^2} = F_y(y,z) = F_z(y,z)$$

$$F_y(y,z) = F_z(y,z) \implies \text{x-momentum satisfied}$$

Figure 9. Strictly Eulerian Stream Function Approach

$$\frac{\bar{\partial}^2\psi}{\partial y^2} + \frac{\bar{\partial}^2\psi}{\partial z^2} = \frac{b\rho}{\dot{m}v_z}\left[\frac{\bar{\partial}H}{\partial y} - T\frac{\bar{\partial}S}{\partial y} + \frac{n_y}{n_x}\frac{1}{\rho}\frac{\partial p}{\partial x} - v_x\frac{\bar{\partial}v_x}{\partial y}\right]$$

$$+ \frac{\bar{\partial}\Psi}{\partial y}\frac{\bar{\partial}\ln b\rho}{\partial y} + \frac{\bar{\partial}\Psi}{\partial z}\frac{\bar{\partial}\ln b\rho}{\partial z}$$

$$\frac{\bar{\partial}^2\psi}{\partial y^2} + \frac{\bar{\partial}^2\psi}{\partial z^2} = \frac{b\rho}{\dot{m}v_y}\left[-\frac{\bar{\partial}H}{\partial z} + T\frac{\bar{\partial}S}{\partial z} + \frac{n_z}{n_x}\frac{1}{\rho}\frac{\partial p}{\partial x} - v_x\frac{\bar{\partial}v_x}{\partial z}\right]$$

$$+ \frac{\bar{\partial}\Psi}{\partial y}\frac{\bar{\partial}\ln b\rho}{\partial y} + \frac{\bar{\partial}\Psi}{\partial z}\frac{\bar{\partial}\ln b\rho}{\partial z}$$

$$\nabla H - T\nabla S = \frac{1}{\rho}\nabla p - \frac{1}{2}\nabla v^2$$

$$\frac{\bar{\partial}H}{\partial y} = \frac{\partial H}{\partial \Psi}\frac{\bar{\partial}\Psi}{\partial y} + \frac{\partial H}{\partial \phi}\frac{\bar{\partial}\phi}{\partial y} \qquad \frac{\bar{\partial}S}{\partial y} = \frac{\partial S}{\partial \Psi}\frac{\bar{\partial}\Psi}{\partial y} + \frac{\partial S}{\partial \phi}\frac{\bar{\partial}\phi}{\partial y}$$

$$\frac{\bar{\partial}H}{\partial z} = \frac{\partial H}{\partial \Psi}\frac{\bar{\partial}\Psi}{\partial z} + \frac{\partial H}{\partial \phi}\frac{\bar{\partial}\phi}{\partial z} \qquad \frac{\bar{\partial}S}{\partial z} = \frac{\partial S}{\partial \Psi}\frac{\bar{\partial}\Psi}{\partial z} + \frac{\partial S}{\partial \phi}\frac{\bar{\partial}\phi}{\partial z}$$

Steps in Iteration:

- Calc 2-D incompressible, irrotational Ψ & ϕ distributions,
- Update Ψ distribution retaining ϕ, v_x, $\frac{1}{\rho}\frac{\partial p}{\partial x}$ & $\frac{\bar{\partial}v_x}{\partial y}$,
- Update ϕ distribution retaining Ψ, v_y, $\frac{1}{\rho}\frac{\partial p}{\partial y}$ & $\frac{\bar{\partial}v_y}{\partial x}$,
- Terminate on constant v_z from one iteration to next,

in the flow field by means of two-dimensional calculations. When updating one stream function, the nodal distribution of the other stream function and all the transverse velocity and derivative components are retained from the proceeding calculation.

Summary:

Three-dimensional, inviscid flow plays an important role in turbomachines. Current state of art makes a quantitative assessment of even the inviscid components of 3-D flow impossible. Research is required to rectify this inadequacy. Important criteria for selection specific areas of research to fund include not only the importance of the results but the probability of success in accomplishing the objectives. Experience at DDAD has indicated approaches to this problem that appear to have a high probability of producing a computational procedure, which will make possible a practical computation procedure for 3-D internal, flow fields within the limitations of present computer speed and memory size.

Initial applications to turbomachinery design will presumably be correction factors to a basically 2-D design procedure. However, the true value of this approach will ultimately be realized when the computations become sufficiently efficient to base design upon a 3-D framework. In this way the number and magnitude of the empirical design corrections can be reduced to the extent that analytical design procedures can replace the slow and costly experimental extrapolation of proven designs.

List of Symbols

A	Arbitrary constant in AIL calculation. See equation (9).
B	Arbitrary constant in AIL calculation. See equation (9).
b	Stream tube thickness
c	Blade chord
C_p	Specific heat at constant pressure
H	Total stagnation enthalpy
$\vec{n}$	Unit vector normal to stream surface
P	Static pressure
R	Gas constant
S	Entropy
T	Static temperature
V	Absolute velocity
$\vec{v}$	Velocity vector
x, y, z	Coordinate directions
γ	Ratio of specific heats
λ	Relaxation constant
ϕ	Stream function
ρ	Density
ψ	Stream function

List of Symbols (continued)

<u>Subscripts</u>:

1	Inlet conditions
C_r	Critical or sonic throat condition
m	Column index
n	Row index
x, y, z	Component in coordinate direction

<u>Superscripts</u>:

t	Iteration index

References

1. Gage, W. R. and Mader, C. L. "Three-Dimensional Cartesian Particle-in-Cell Calculations," Los Alamos Scientific Laboratory Report LA-3422 (1966).

2. Stuart, A. R. and Hetherington, R. "The Solution of Three Variable Duct Equations," International Symposium on the Fluid Mechanics and Design of Turbomachinery, Penn. State Univ (1970).

3. Wu, C. H. "A General Theory of Three-Dimensional Flow in Subsonic and Supersonic Turbomachines of Axial-, Radial-, and Mixed-Flow Type," NACA TN 2604 (1952).

4. Katsanis, T. "Computer Program for Calculating Velocities and Streamlines on a Blade-to-Blade Stream Surface of a Turbomachine," NASA TN D-4525 (1968).

DISCUSSION

(Bullock) Let us always remember that these rotor data in Figure 1 are some sort of weighted, time-averaged quantities.

(Fagan) Yes, and the three-dimensional effects are the result of the stator wakes in front of the rotor not being completely averaged out by the rotor.

(Lakshminarayana) Then you take the average values.

(Fagan) Yes, they are averaged values.

(Lakshminarayana) How do you get the blade-to-blade efficiency?

(Fagan) The efficiency was deduced from total pressure, temperature, and velocity measurements. For a more complete description of the test and data reduction techniques, I would suggest NASA contractor's report, NASA-CR-1803, "Design and Experimental Results of a Highly Loaded, Low Soliditary Tandem Rotor," by J. L. Bettner of Allison.

(Katsanis) That's not completely general, because you have a constraint on your corner. You are assuming that the streamlines follow the corner of the duct, right?

(Fagan) There is a lot of trouble handling the accounting system, but in principle the streamline can change from that surface to that surface when it goes around the corner and vary locally in the corner. It is like the argument about the cascades; we don't do the walls well. We will try that again. But the streamlines can intersect on that surface or that surface as it rotates, if you keep the accounting systems appropriately.

(Katsanis) This system will allow the stream surface to actually twist and change from a vertical to a horizontal one.

(Fagan) There are obvious problems in this Lagrangian approach that you have in any programming approach. When you start out with a coordinate system that is horizontal and end up with a surface that is vertical, there are problems.

(Katsanis) I am not talking just about the problem of doing it but conceptually.

(Fagan) Conceptually, I believe it is possible.

(Katsanis) I don't understand what the iteration procedure would be that would enable you to do this. In other words, if you start with some assumptions like this of stream surfaces, how do you get from an initial orthogonal set of stream surfaces to one that starts twisting around?

(Editor's Comment) Dr. Fagan suggests that the answer to Dr. Katsanis' question is best given in the part of his written text concerning the communication problem.

(Smith) Why did it warp, was there something going on in the direction perpendicular in this plane?

(Fagan) Yes, it came in with the total pressure distribution. Basically you can think of it as being in a rotor, although this isn't in a relative coordinate system at this point.

(Smith) In your example here, what did you assume to make it work?

(Fagan) What did I assume to make it work? What I have chosen is a normal pressure distribution, which would be a source of its warping, but I have not tried to close between the magnitude of that and the amount of warpage we put in.

(Smith) O.K.

(Fagan) It is a demonstration problem; I freely admit. It will only tell us that if I have radial pressure distributions and warped stream surfaces, it changes from what I would get from a 2-D problem. It is the only effect I would want to see out of this. In Figure 7 the solid line is the pressure on either the I.D. or the O.D. of this channel. One other point about the channel, the straight section beyond here is a very slightly turning blade section. Part of that is to make sure that the problem would converge when we did it, because it is a new problem. And it is not only with convergance that we have problems many times, but also that was a straight section set up so that the static pressure in the trailing edge would be balanced as if we had a repetitive boundary condition at that end. In the straight, unwarped condition then, those directions are chosen such that the static pressure repeats from here to here and all along there (Figure 7). We now come in with a ten degree warp, that means that it was flat here and warped ten degrees by the time it got here and it continues at the same rate of warp then on out the edge and the cross radial, if I can call it that, pressure distribution parameter. This is pounds force per pound mass; it is comparable to about 1 psi per inch. It is not particularly high for the zero-warping case. For a 10-degree warp with 10,000 pounds force per pound mass, the transfer pressure distribution shows that we had signigicant differences in the surface velocity distribution at different locations on the blade also, to a certain extent, but that doesn't make any difference if it were truly a two-dimensional problem. Furthermore, we find that for the exit conditions now, we no longer get this repetitive condition, which is physically possible because they are not at the same radius, if I can call it that, even though they are repeating in angle and the flow calculation is repeated

(Smith) It is okay, but is it a reasonable thing to look at? Are you going to learn anything from that?

(Mikolajczak) You appear to have an unrealistic pressure gradient?

(Smith) Yes, you are implying a very large spanwise pressure gradient at discharge but none at inlet or very little at inlet, and I don't think this is the kind of case that you . . .

(Huffman) He is not claiming that this is realistic, only that it is a true challenge to the method, and obviously when you challenge a method you want a more difficult case.

(Lakshminarayana) You are not satisfying the boundary conditions.

(Fagan) This is a guided channel. Not meant to be a blade row, but I draw the analogy with the blade row.

(Lakshminarayana) You are introducing a straight section, aren't you? You cannot have a pressure gradient at the entrance.

(Fagan) That is the same as if we were to take in the 2-D sense the blade row and put it together as a guided channel. It has got repeatability conditions across here and across there, and a nice flat stream surface going through it, so it represents in a 2-D sense that guided channel that we would want to make the analysis for. Now, as long as there is no warpage of that stream surface, it matters not at all how much radial or cross pressure gradients you have. Because that only comes in either y- or the z-momentum equation if you've got it until that surface. Now, let's tilt the surface a little bit, and admittedly the numbers are large to demonstrate the point; but it does show that if you tilt the surface with transverse pressure gradient, you are going to get a different answer. Now, that is not the true boundary condition necessarily, but you can't say that until you get to the 3-D flow problem. They aren't at the same radial location. They could be different. In principle I would guess they would be different, from one side to the other.

(Smith) It is kind of hard to visualize that there is a case where that could happen to you. I suggest perhaps a case where you have a uniform core far upstream, and then you have a blade row that is very highly twisted so that the circulation is very non-constant along the span, and this would be a more interesting case for comparison. There you can get some large warping of the surfaces that you are looking for and still have reasonable boundary conditions. But for that case you won't get large gradients of static pressure downstream without . . . I kind of think, from looking at your pictures, if you remove that large downstream pressure gradient and sort of squeeze your pictures together, that there wouldn't be so much difference anymore.

(Bullock) I think we have to know where the pressure gradient comes from.

(Louis) Do the surfaces continue to warp downstream?

(Fagan) The surfaces continue to warp downstream. They are warping continually at a rate that gives you 10 degrees or 20 degrees or so and continues to go on downstream. That's true. Once you start that warping process, it will continue.

(Bullock) Is the gradient of total pressure perpendicular to the plane of your chart?

(Fagan) The pressure gradient is perpendicular to what would be the plane in a 2-D sense.

(Bullock) In this case would it be perpendicular to the stream?

(Fagan) Yes, it would.

(Bullock) Your conclusions are qualitatively correct. As the flow progresses downstream, the static pressure normal to the stream changes. We thus have pressure gradients perpendicular to the stream. At the same time, the turning of the stream also requires pressure gradients normal to both the stream and the gradient in total pressure. These gradients cause each individual stream tube to twist; the amount of twisting depends, among other things, on the gradient in static pressure downstream.

(Fagan) Yes, I don't set the static pressure downstream.

(Bullock) As the incoming velocity increases, these aberrations should increase.

(Fagan) In essence that is what I am saying. As it goes through, these are different, if I can make the analogy with the blade row now, these are at different radial locations on the top and bottom of the channel so the repeatability conditions per se do not apply. You have to solve the full 3-D flow field and have to repeat at the same radial level admittedly, and that is a good boundary condition but only in a 3-D sense. This surface now, since this point translated one thickness here, does not end up at the same point as that one.

(Fagan) This one is truly a flat surface in a guided channel.

(Mikolajczak) Is it a fixed geometry surface that rotates?

(Fagan) Yes, it is always flat. It only rotates.

(Bullock) It has to be, because of the vorticity in it.

(Fagan) It is a crude simulation of the vorticity effect.If you have a gradient from bottom to top of the total pressure coming in, you will see this kind of rotation. It's like a Squire and Winter kind of analysis. It is an oversimplification in that when you look at that it won't turn out to be a plane. The only point I can draw with this is that that's a 2-D problem, and in 3-D it does demonstrate that you can get differences in simple things like static pressure distributions on the blade rows.

(Smith) I guess it would be more convincing for us intuitive engineers if you had a case that we could really visualize that that's what is going to happen in this case.

(Fagan) That's the next step.

(Mikolajczak) In a blade row you have to satisfy some periodicity condition. The apparent disregard of this condition in your example causes some disagreement.

(Fagan) The periodicity condition is real and has to be satisfied. The only argument I made about the downstream conditions were that they don't have to be satisfied on that stream surface. Because after this twist, the top and bottom of the stream surface aren't at the same radial location anymore. This point has to be periodic with its neighbor but not with this end of the calculation, because it is not at the same physical location in space.

(Smith) On that point, Alex, I think he is right. We don't have to match conditions. The thing that bothers me is that conditions are so far different in discharge static pressure for such a little bit of turning in a cascade that I can't conceive that this is even close to a real case.

(Fagan) You agree that you have to look at 3-D effects, but we don't agree on the magnitude yet.

(Smith) Right.

(Fagan) I can't make a convincing argument on the magnitude at this time.

But it does tell you that there is a significant difference that you see only by letting the stream surfaces rotate significantly. You have to strike that. There is a difference that you can see right through the blade cascade if you consider 3-D effects. You never know that until you solve the 3-D flow problem.

(Smith) You will find other cases where there is a significant twisting or warping of the stream surfaces and yet the 3-D effects are not very large. I think the case I cited, if I take irrotational flow coming into twisted inlet guide vanes. If these are relatively high aspect ratio inlet guide vanes and if the circulation is strongly non-constant, I can get significant warping of the surfaces as they come through to get the velocity distribution you get around the profiles except right at the end walls, very similar to the potential case or the 2-D case. If you persevere and get calculations in such cases, I predict that that's what you will find.

(Fagan) But you said the inlet conditions were the same.

(Smith) Well, yes, I have.

(Fagan) But if you have variable inlet conditions because you have variable radial distribution of losses from the bottom to top, then when you see this rotation you are bringing low energy fluid into a region where you thought was high energy fluid. In your case you get this rotation, and the third that is rotating is the same kind of fluid (i.e., from the part that is replacing); and you might just as well say that it isn't rotating. But, if you have variations of inlet conditions and from one side to the other and this rotation takes place, then you are going to see the significant effect even so. If you guarantee that the total pressure coming in to the blade row is constant from top to bottom, then it doesn't make any difference; but if you had non-uniform velocity or non-uniform work conditions in upstage rotors, stators then I think it can be very important.

(Katsanis) There might be another reason why you get such a large difference. Perhaps, you have this warping, the stagnation streamline may shift considerably from the initial zero-warping conditions. In other words, you are assuming the same channel all the time, and then when you try to get a periodic condition this may not be a correct location.

(Fagan) I can only argue that this is the right channel for this 2-D problem.

(Katsanis) Yes, that's right.

(Fagan) We do agree that it has to be at least a different channel for the 3-D problem. Again you come to the conclusion you have got to look at the 3-D effects.

(Bullock) Isn't this your principal message -- that your techniques yield a solution which converges for these boundary conditions?

(Fagan) I was also trying to convince you that it was important to look at these things.

(Bullock) One of our important conclusions of the previous hour was that three-dimensional effects were important.

(Oates) I am wondering if you mean that if you don't look at the 3-D effects, you won't find them. In other words, if you go into the 2-D analysis and say it works, you can never be suspicious.

(Fagan) When you run the machine you are then going to get suspicious, I guess, but what I meant is if you take a point of view of design that says you want to look at the flow field. You don't always take that point of view in the design. You take vector diagrams in, vector diagrams out and maybe partially look at the flow field by meridional calculations, or you may even put in one blade-to-blade calculation but on an axisymmetric flow surface; you then say that we can read out the flow field sufficiently to do design.

(Oates) I was wondering if the Squire and Winter form of calculation would shed some light as to whether the 3-D effects are significant. Do you feel you have to go to the full 3-D calculations to find out whether they are significant?

(Fagan) That is not a proven point. You can use Squire and Winter literally and tell how far you think that twisting went through. I don't believe it is right. That is my only answer to that, and I can't prove that or disprove it. It seems to be independent of the Mach number. It is the relative change in total pressure that dominates the Squire and Winter situation. You get the same relative change, bigger in absolute value at a Mach number of 1.2, you had at a Mach number of .2. You will get the same rotation. Physically it doesn't seem as if that is probably the case. Well, simply knowing how much the stream surface rotated, what does that tell you about the pressure distribution say on the plate? Do you have ready information on that? I still have trouble making up the pressure distributions, although you can follow that up with some additional logic to make estimated velocity distributions on a plate. But that's kind of hazy too or at least it is hazy for me. Just knowing how much that stream surface rotated is not the end of the course, and it is only a starting point. I am saying you should use a 3-D calculation in order to find out how much it has rotated, instead of Squire and Winter's. That is one point, but the next point is you also get the static pressure distributions, velocity distributions which you still don't get out of Squire and Winter's and Hawthorne's analyses.

(Katsanis) What do the velocity distributions look like at the end of the channel? Do you see anything like the corner vortices at the end of the channel?

(Fagan) Again this problem couldn't do that. This is a two-dimensional problem; it's a relaxed two-dimensional problem worked on a warped surface, but it is still a 2-D problem. I must admit we've tried to convert the 3-D problem in the very first way I showed you just using x-momentum equation and the y-momentum equation. We discovered through what I termed the communication problem the simple y-momentum equation is not going to recognize any plane surface, which is the easy case to start with, so you have to do something about it. If you can put in an answer that is roughly right, and there you might use Squire and Winter, we just haven't gotten that far. Put in an answer that is roughly right and I think it will converge, because there is a possibility of communication and you just can't work from a long ways away from the right answer.

(Unidentified) How much of an angle could you turn the flow through and still have your system converge?

(Fagan) If I use what I call the Lagrangian approach that I started with, you can't get to 90 degrees, or I am in trouble. It might work in a blade row, but it is not going to work for a turning-duct kind of problem. But I think that is just an incentive to set it up as an Eulerian kind of problem. Then it can converge on any of these situations. I don't see any stumbling block to it converging.

(Lakshminarayana) Are you familiar with the work by Marsh and Smith, the matrix-through-flow solution? How does your solution differ from theirs, in terms of the numerical technique?

(Fagan) There are two differences that I know of. They have used higher order differencing, so they could spread the meshes; and they have used matrix methods to solve the Poisson equation, as best I know. So those are the two differences that I know of. In principle they are very much alike. I have never seen them publish any data, although in principle you could, that wasn't either on a radial plane or an axisymmetric plane.

(Smith) I think that is right. He did present solutions on both surfaces actually. But I gathered from what he said, that he had not really gotten yet to the problem of iterating and getting a convergence. And furthermore, I thought he sounded a little pessimistic about the prospects of ultimately succeeding in doing that.

(Fagan) Well, I think this talk has got some pessimism in it, in that I haven't accomplished it. It has got some optimism in it, in that we have found at least three ways that we are in the process of trying out the mixed flow.

(Lakshminarayana) How do you intend to satisfy the Kutta condition for the blade in three dimensions?

(Fagan) I would like to work in ducts for the time being. But I think the answer to that would be a trial and error approach at locating the stagnation streamline. That is not going to be an easy thing to do. If you go the Eulerian approach that I am talking about, you have a repeatability condition at the back. You are talking about the same physical location on those two streamlines, and you apply the repeatability. If I really take the warped surfaces, then I still have to use the repeatability condition; but I have to jump between different surfaces to know. You can't apply it all in one calculation. I have to do the multiple phi surfaces before I can know that I did or did not have repeatability. You are kind of asking me to dream, but the thought is that we can do this, and do the boundary layers and the wake calculation, and put the whole thing together. That is something else. As I say, I really talk in terms of duct flow; I have to admit that at this point in time.

(Vavra) One could also ask, what else do you do, leave the situation as it is? You take the through-flow conditions axisymmetric, and then you establish the blade-to-blade solution on assumed axisymmetric surfaces. That is where we stand today. That has its limitations, and I think it can't give us all the answers which we need. Or we do it very crudely; and think, for instance, that an axisymmetric surface tends to become a cylinder. Then you have no differences any more between the flow in a stator and the flow in a rotor. That is one of the troubles which you run into.

(Katsanis) Your technique would be no particular problem in extending it to a rotor. You haven't included the necessary terms; but a rotating field, I don't think, would be any more difficult. You get a different solution definitely, but it would be no essential problem.

(Vavra) The difficulties are large, we all realize this. But here is the point that with one method you cannot proceed any more, and you try to do it this way. Does anyone know another possibility to do this a better way?

(Katsanis) I don't feel as if we've gotten all we can out of two-dimensional solutions yet. At least I'm still working on them. For example, in two-dimensional solutions, you talk about blade-to-blade and meridional solutions. However, there are three two-dimensional planes you can work in. The third one is a cross-channel two-dimensional plane. At least in turbomachine design, there have been programs that work in this plane, in other words, just consider the gradients from hub to tip and blade to blade based on simple assumptions, like in a well-guided passage. This has been the basis for designing axial-flow turbines for a long time. This basic idea can be extended really to fairly general cases, which I've recently done. There will be a T.N. out on this real soon. This will be NASA TN D-6177. It is a very simple concept, and you are limited to very well-guided channels. However, the method is very useful, particularly where you are interested in obtaining choking-weight flow at a throat and you have important variations both from blade to blade and hub to shroud; it seems to me that this is a useful way of getting at it. This is approximate, but because of the important variations at near sonic velocities, you want to look at the variations in both directions simultaneously; not just blade to blade and hub to shroud separately, but in one calculation be able to go hub to shroud and blade to blade. And you can do that based on simple assumptions.

(Vavra) The point was made, what do you do with the Kutta condition?

(Katsanis) For blade to blade, I found that the Kutta condition is not useful for obtaining the stagnation point.

(Vavra) I don't mean directly the Kutta condition, but how does the flow take off of the rear part of the blade?

(Katsanis) The criteria that I found to be most useful is to just by trial and error assume downstream flow angles, and then see what velocity distribution you get on the blade surface. You choose an angle such that the two velocities match, where they extrapolate to the free-stream velocity. You have a better handle on this, especially with a curved trailing edge or even if it is sharp. It becomes very sensitive.

(Vavra) You see this best on the field plotter. We just do this electrically, and then have to put in circulation because we do this without circulation. The location of the rear stagnation point is awfully critical. Then we do something like you said. We guess the angle and work back to the trailing-edge stagnation point. This is always associated with error, because it makes a lot of difference just where you put it. You said what you think you can still do with two-dimensional methods or combinations of them. Where do we go from there?

(Katsanis) First of all, the two-dimensional methods, I think, are always going to be very important to us because certain types of geometry are more two-dimensional than others. If you have a very high hub-tip ratio and very little change from hub to tip, two-dimensional methods will be pretty good; or if you have a geometry where you have very high solidity but maybe big variations from hub to tip, then meridional plane solutions give you good answers. However, in certain geometry, like in centrifugal impellers, the two-dimensional methods are hopeless. That is where we really need three-dimensional methods, and that is also where it is going to be the toughest to get. Even with what Fagan is doing, it is going to be tougher when he tries to get to those types of geometries.

(Fagan) One thought about the exit conditions, we have written this program in such a way that it handles the center body. We haven't tried all the possibilities for doing this, but if you use the channel as the two outside blades and the answer that you want is the center blade, you can then feel that you don't have to be as accurate in setting the exit conditions; but rather you read the flow directions. You go to the cascade kind of flow.

(Lakshminarayana) Yes, but you still have to fix the streamlines of the two outside blades.

(Fagan) But the streamlines from outside blades being operable is going to be less sensitive than having to fix the exit streamlines from the blade you want to get pressure distributions off of.

(Lakshminarayana) I'm sorry, I don't agree with that. It is as sensitive; after all the streamline is going at the same angle on the three blades.

(Fagan) It doesn't have to. Put repeatability on the first and third blade, and the center streamline will go in whatever direction it wants to. I'm just thinking that is a way to take some of the sensitivity out of the problem. You still probably have to iterate. Then you use that direction on the first and third. It does appear that, in doing a blade-to-blade problem, it might make sense to run two blade passages rather than one.

(Katsanis) I think that is a good first step to three-dimensional flow. I think there is still going to be a long way to go. It looks to me like the best thing I've heard of for true three-dimensional flow.

(Serovy) I think you've got to have feelers like this in this direction. What he is doing is okay, because you've got to keep pushing out a little way in that direction. Try it and see what happens. Like Katsanis, I would hate to see the so-called two-dimensional classical methods completely eliminated now, because we would be dead in terms of current and reasonably near-future designs.

(Fagan) My view of where it would fit in the design procedure is after you have done everything you have already done, and you take another pass with this.

(Serovy) Yes, I'm sure that it would work in that way, because you wouldn't be able to make your type of calculation too many times.

(Fagan) Well, discounting the time, it is still an analysis and not a design procedure. There are all sorts of reasons that you have got to have what you think you want before you can apply this. I think it would be useful to go that one further step in the design, as soon as this were here. I don't know whether I would call that a basic framework or a design or not, but in a sense it would be.

APPLICATION OF RESULTS OF RESEARCH TO ENGINE DESIGN PROBLEMS

Discussion Leader: Dr. R. E. Henderson

Presentation: Axial Machinery Development Trends

by

Mr. J. W. McBride

GAS TURBINE RESEARCH FRONTIERS*

by

R. E. Henderson

INTRODUCTION

The history of the gas turbine engine shows that this type of engine has been developed very rapidly. The main effort has been concentrated on the engineering development of larger engines and diversified engine types, i.e., turbojets, turboprops, turboshafts, single shaft or multi-spool, fixed and variable geometry, low and high bypass ratio fans, and free or fixed turbines. The historical background has created an illusion that an extensive fundamental technology data bank exists which allows this rapid progression. In actuality, development of each engine has depended upon past empirical knowledge with a strong "time-oriented" engineering attack to obtain "fixes" for erupting problems. These "fixes" had to be compatible with the existing engine hardware for the particular model being developed. Consequently, there has been a minimum of technology transfer from one engine design to a succeeding design. The knowledge learned on one engine tended to be peculiarly oriented toward that mechanical configuration. As a result, the time, manpower and money to produce each "new" engine is increasing rather than decreasing. One consequence of this increased cost is that there has been an abandonment of concurrent development of alternate engine designs. The development of only one engine thus results in a high risk program. The Armed Services' response to reducing the risk has been (1) to require a demonstrated gas path before an engine development program is committed and (2) to require a demonstrated engine/airframe compatibility.

This is not to say that fundamental technology does not exist and that no effort was expended on research during the past quarter century. Concurrent with the above cited engine development, extensive gas turbine technology was developed. However, the normal gas turbine engine requirements generally exceed the mathematical expressions representing the technology. As an example, gas flows are analyzed with 2D cascades; however, the "real world" of rotational flow through a compressor is much more complex as shown in Figure 1. Technology has given qualitative insight into the phenomena occurring but has not yielded universal quantitative results to guide an engine designer. Therefore gas turbine technology and gas turbine design has coexisted in adjacent worlds without too much cross pollination. This same division between the pure world of technology and the complex, real world of engine design as cited above for fluid mechanics also exists in other fields such as heat transfer, noise, and combustion. The interactions between disciplines or within a specific discipline are generally more complex than can be recognized by current technology, especially when one considers a normal range of operating conditions. Figure 2 illustrates the idealized progression of effort for an engine development program. In actuality, the wind tunnel and rotating stage activity shown in Figure 2 is divorced from the main thrust and lacks direct support of the Armed Services. This void represents a challenge to the Researcher to make his efforts more relevant.

*Editor's Comment: This is the paper on which Dr. Henderson based his introductory remarks.

DETROIT DIESEL ALLISON APPROACH

Allison Research is attempting to accomplish an idealized solution to the internal flow problem in an engine. The ultimate objective is to develop a design synthesis capability which will define the complete flow path (number of stages, solidity, type of blade, blade row spacing, blade length, clearances, blade shape from hub and tip, etc.) from a minimum statement of the requirements (such as overall pressure ratio, airflow, max. diameter). Accomplishment of this idealized objective requires the following:

- an understanding of the phenomena involved over the entire operating range,
- the capability to prepare an analytical model or a mathematical model emphasizing these physical principles,
- the capability to conduct critical experiments,
- a reiteration between experimental data and mathematical model to produce a workable design system.

The benefits accruing from such an approach are to produce relevant results with a minimum of funding. In turn this will minimize the risk involved in building high cost turbine or compressor rigs. The ultimate net result will be a major advancement in the state of the art by integrating all the phenomena occurring in a flow field.

ADMIRAL HOLMQUIST'S PRESENTATION

The following sections of this discussion will be confined to those projected research areas closely related to the listing cited by Admiral Holmquist. His topics form an excellent base for submitting comments. Since the Monterey conference emphasized fluid mechanics, the following discussion will principally relate to fluid mechanics and will exclude such items as metallurgy, controls, and manufacturing research. However, similar discussions in those areas with the same format would be applicable.

1. Stoichiometric Engine

This engine presents a challenge on many frontiers of technology. The three main research areas relate to (1) combustion, (2) flow analysis, and (3) heat transfer.

- Combustion -- Research is needed to supply technology related to designing a combustor which (1) will operate over the entire f/a spectrum, (2) can be ignited over as wide a range as possible, (3) can produce an acceptable outlet temperature pattern, (4) can keep the burner liner cool, and (5) can avoid combustion instability.

- Flow Analysis -- Research is needed to supply the technology related to losses incurred by injection of the blade cooling air into the main stream. This is a strong phenomenological-oriented task involving skin friction losses, mixing losses, jet deflection losses and heat transfer.

- Heat Transfer -- Blade cooling is one of the foremost problems which is intertwined with internal flow analysis in individual blade segments with combined types of cooling (convection, impingement, conduction, radiation) and blade cooling configurations (jet flap, slots, holes, porous, Lamilloy).

2. Flow Analysis in Large Compressors and Turbines

A flow analysis effort has been outlined briefly in the introduction; Figure 3 summarizes the approach. Figure 4 indicates the current envisioned task structure of the 2D and the 3D portions of this problem. As indicated on this figure, technology exists for some tasks; in other task areas, work is in progress, proposals have been submitted to accomplish the subtask, or the subtask is dormant. We are currently negotiating with NAVAIR on a Three-Dimensional, Inviscid Flow Analysis which proposes that by the use of an iterative procedure, wherein a series of mathematical, two-dimensional flow calculations are made, it is entirely feasible to build up three-dimensional flow fields.

3. Materials and Fabrication Techniques

New materials currently under development (namely, composites for fan blades and ceramics for turbine blades) offer the promise of uncoupling the gas path from the mechanical limitations of the blading. Thus gas path improvements or changes can be considered which was not possible with old materials.

4. Blade Tip Losses

This loss is one of many involved in the flow analysis cited in Item 2 above and illustrated in Figure 1.

5. Inlet Design

The flow analysis program briefly outlined above will be applicable to flow in ducts as well as rotating machines. The Three-Dimensional, Inviscid Flow Analysis cited in Item 2 above will supply a technological base for solution of this problem. Dr. J. R. Fagan discussed this approach at the Workshop. Enclosed is a copy of his paper, "Three Dimensional Flow Analysis in Turbomachinery."

6. Reliability and Maintainability

With the advent of large fan engines, the aeroelastic phenomena associated with the long, thin, high aspect ratio, cantilevered fan blades have come to the fore. To obtain long life, lightweight blades require the blending of the aerodynamic and the elastic or structural forces into a unified concept. Allison has done extensive analytical work to develop such a unifying mathematical concept and is currently proposing to extend this work with laboratory experiments. Recently we submitted a program to ONR (thru Project Squid) to accomplish this activity.

7. Sheet Metal Turbojet

A low cost engine is an admirable objective. Since the gas path engine parts account for over 50% of the engine cost, any cost reduction effort must be based upon a thorough understanding of flow analysis. The first low cost engine will probably be a small sized engine. Our viewpoint has favored a centrifugal compressor for such a unit. The internal flow passage in the compressor and in the diffuser are strongly dominated by boundary layer effects. When we fully understand and know how to account for boundary layers, we will be closer to knowing how to make a low cost centrifugal compressor a practicality.

Dr. G. D. Huffman of the DDAD Research Staff has directed his attention to the boundary layer phenomena. At your recent Turbomachinery Workshop, he presented a paper on boundary layers. A copy of his paper, "A Re-Examination of Some Retransitional Flows" is attached.

8. Re-Engineered Engines

The potential to redesign an existing engine and thus reduce cost and achieve simplicity exists. In fact, the design concepts of any engine can be extended to either larger or smaller engines. New engines for the military require a redesign because simultaneously with the change in size are revised engine goals (generally higher specific power, lbs thrust per unit airflow; lower specific weight, lbs weight per unit output; and lower specific fuel consumption, lbs of fuel per unit output) which requires an advanced state-of-the-art design incorporating a new flow path.

The subject of a "re-engineered engine" needs considerable expansion as it is closely related to the armed services "needs." Traditionally an engine is developed for a specific application. Once an engine is designed, it generally has a limited number of additional applications. For each new application, funding is generally restricted to that associated with the peculiarities of each installation. Funding has not been allocated, nor justified, to encompass the attainment of the fundamental gas path knowledge to support a redesign. Further the CIP (Component Improvement Program) funds associated with a normal production run of an engine does not encompass this type of activity. Basically an engine development is restricted to the armed services funding needs. After an engine has been designed and developed, the funding "needs" relate more to amortizing the high development cost over further production (thus reducing the development cost per engine) than to increasing the development cost by going into more technical depth after the engine has been developed. Further, the armed services needs for future engines have related to bigger and better aircraft rather than aircraft which can utilize a redesigned engine. Thus the total turbine engine industry has a strong revolutionary orientation rather than an evolutionary "flavor." An evolutionary approach more nearly matches the "re-engineered" engine concept than does the revolutionary approach. This whole problem is further aggravated by the increased effectiveness of modern weapon systems which require engines in quantities of hundreds rather than thousands. The requirement now is to design the 1972 Cadillac without the benefit of the preceeding yearly models.

9. <u>Internal Instrumentaion</u>

The traditional approach to this problem would be to utilize a maze of instrumentation which essentially duplicates engine test-stand instrumentation and, by means of a computer, accomplish the desired diagnosis of the engine's "health." This concept involves measurement of a gross parameter, such as vibration, whereas a failure inducing phenomena (such as loss of a blade) preceeds this gross parameter. Failures in the gas path may be precursors of more extensive engine failure conditions which could cause the aborting of a mission. Again the fluid mechanic knowledge gained in defining the gas path can be of assistance in the definition of the critical parameter to measure. Such knowledge can also limit the number of parameters to measure.

10. <u>IR Signature</u>

Allison has expended considerable effort in developing Lamilloy for use in high temperature areas of an engine. One of the significant advantages of this Lamilloy is that the parts in the "hot sections" are kept cooler with a consequent lowering of the engine's IR signature.

11. Noise

The universality of the emphasis on fluid mechanics is illustrated by discussing noise. Pressure gradients such as the cyclic disturbances in the wake of each blade row are major sources of noise. Thus, a knowledge of the gas path and the ability to predict changes in the various parameters can assist in producing quieter engines.

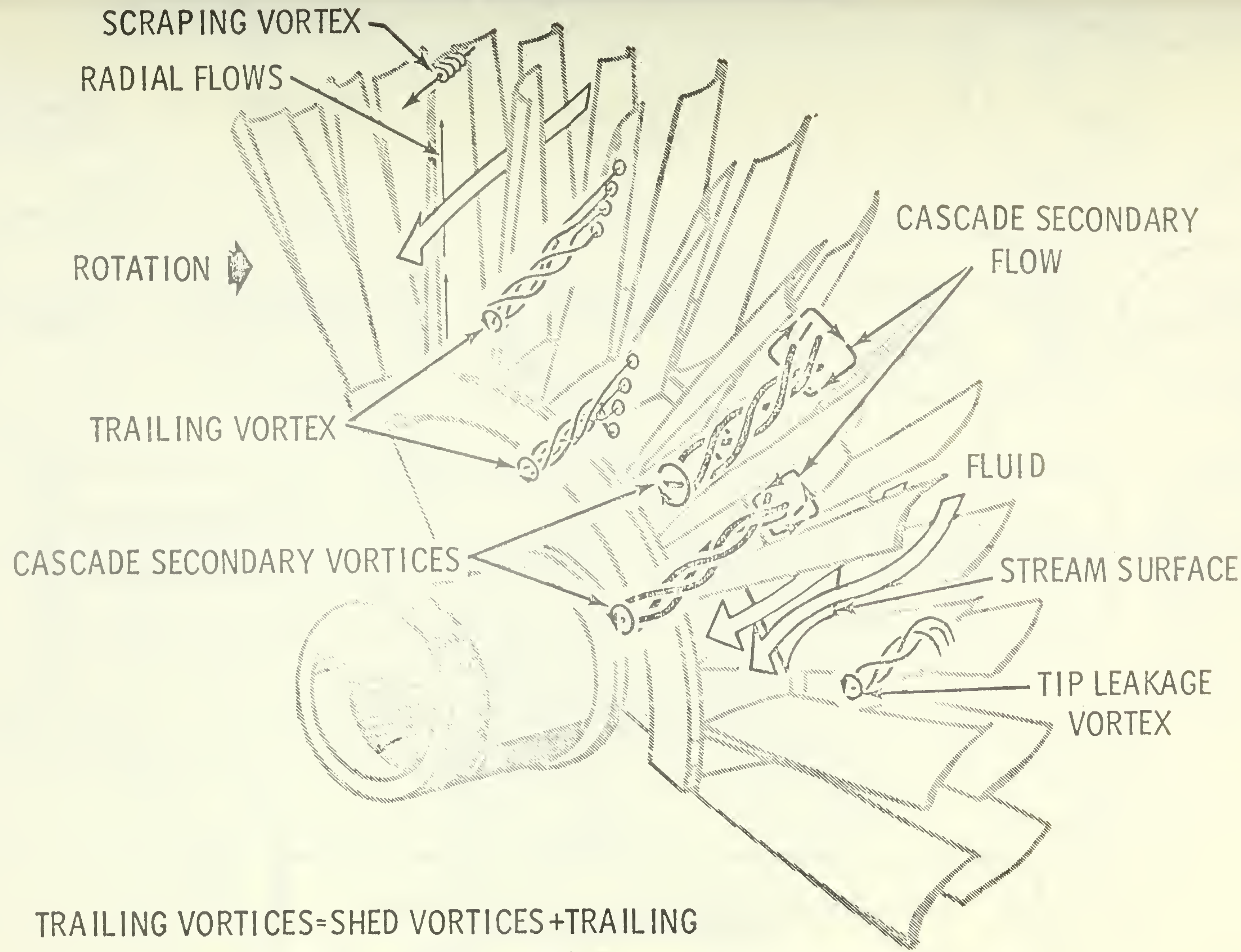
Allison Research has a heavy gas acoustic facility to investigate acoustic phenomena. This facility is illustrated and discussed on pages 6 and 7 of the enclosed brochure.

12. Pollution

Admittedly this topic has limited applicability to combat aircraft. However, the same methodology previously discussed is being applied to this field. A major contributor of knowledge in this area is the increased combustion knowledge expressed in current mixing and reaction computer programs. This program, in conjunction with contemporary knowledge of turbulent viscosity and emission reaction kinetics, has the potential of yielding extensive data for the internal mapping of combustion exhaust fields.

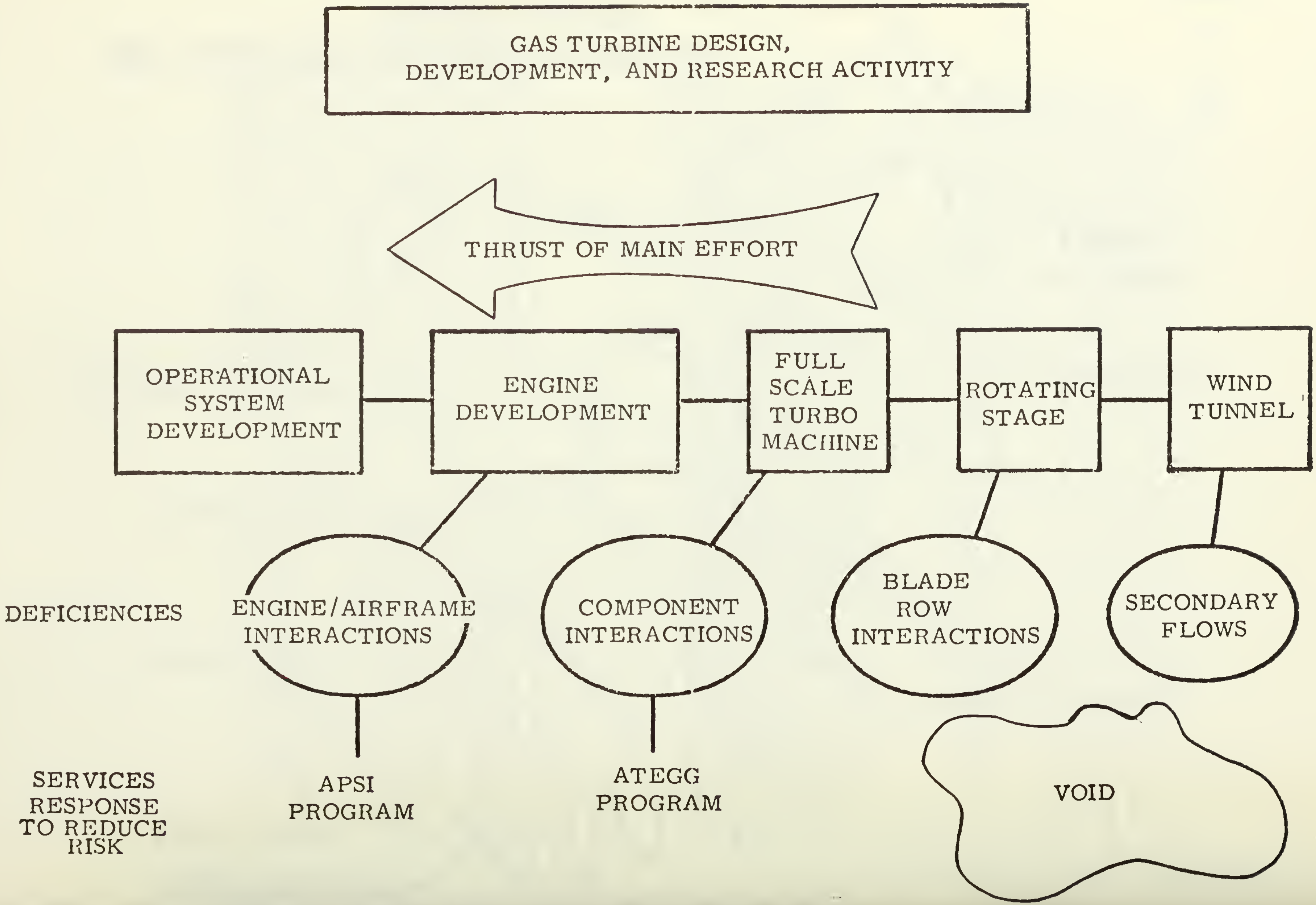
SUMMARY

Past laboratory experiments and scientific investigation have yielded an extensive background of "pure" phenomenological data. All of this activity has resulted in a deep understanding of the internal problems of an engine, but lacking is the direct applicability to engine design techniques. Accordingly, the credibility of research results has suffered in comparison with actual engine test results. With the current computer technology, unifying mathematical concepts are possible to account for the complex simultaneous interactions between the present isolated technologies. Mathematical modeling, supported by critical laboratory experiments, is an extremely low cost method to produce meaningful results. Figure 5 illustrates this concept which offer an opportunity to achieve major advancements in the gas turbine "state of the art" by improving the credibility or acceptability of research efforts.

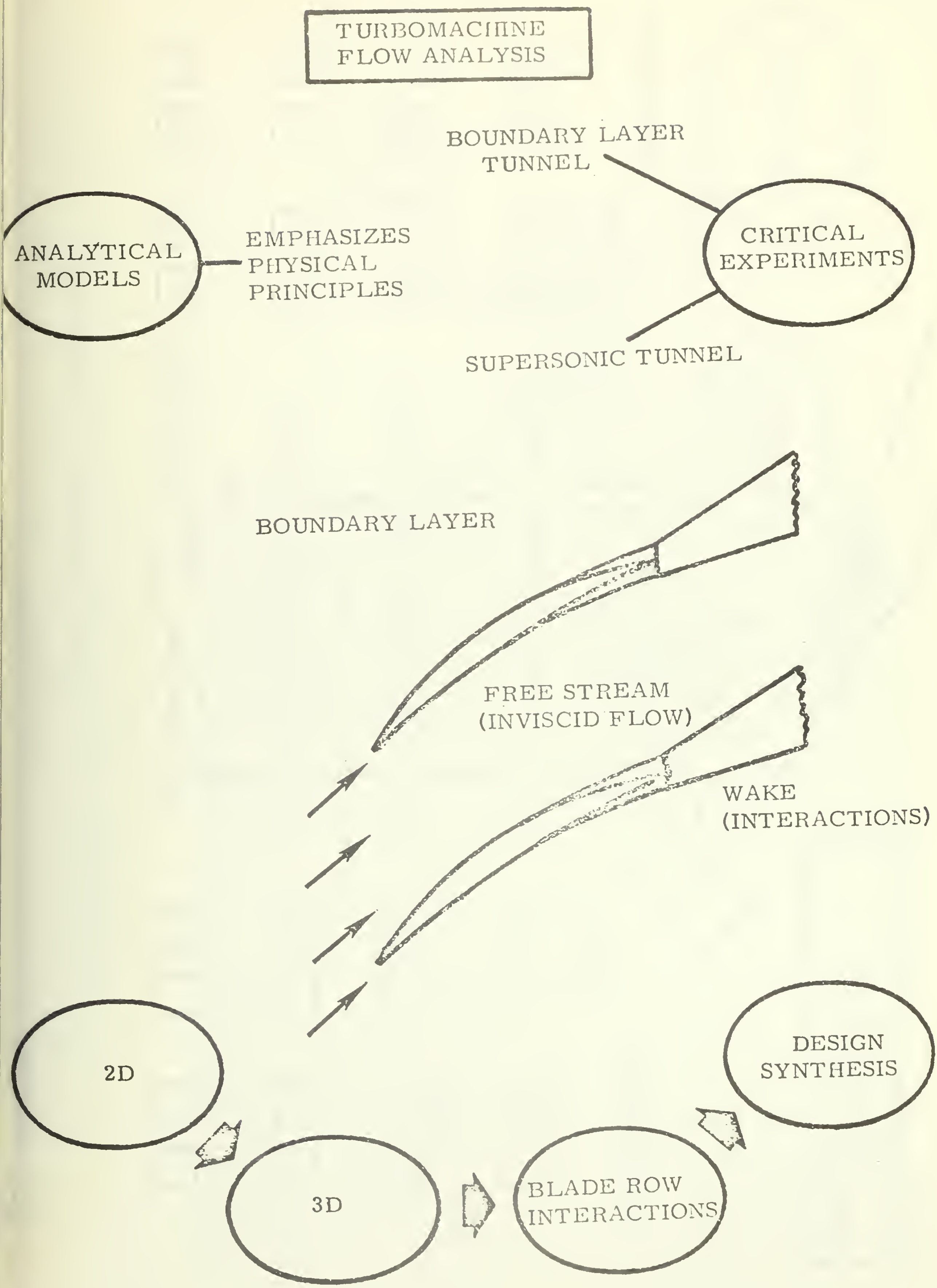


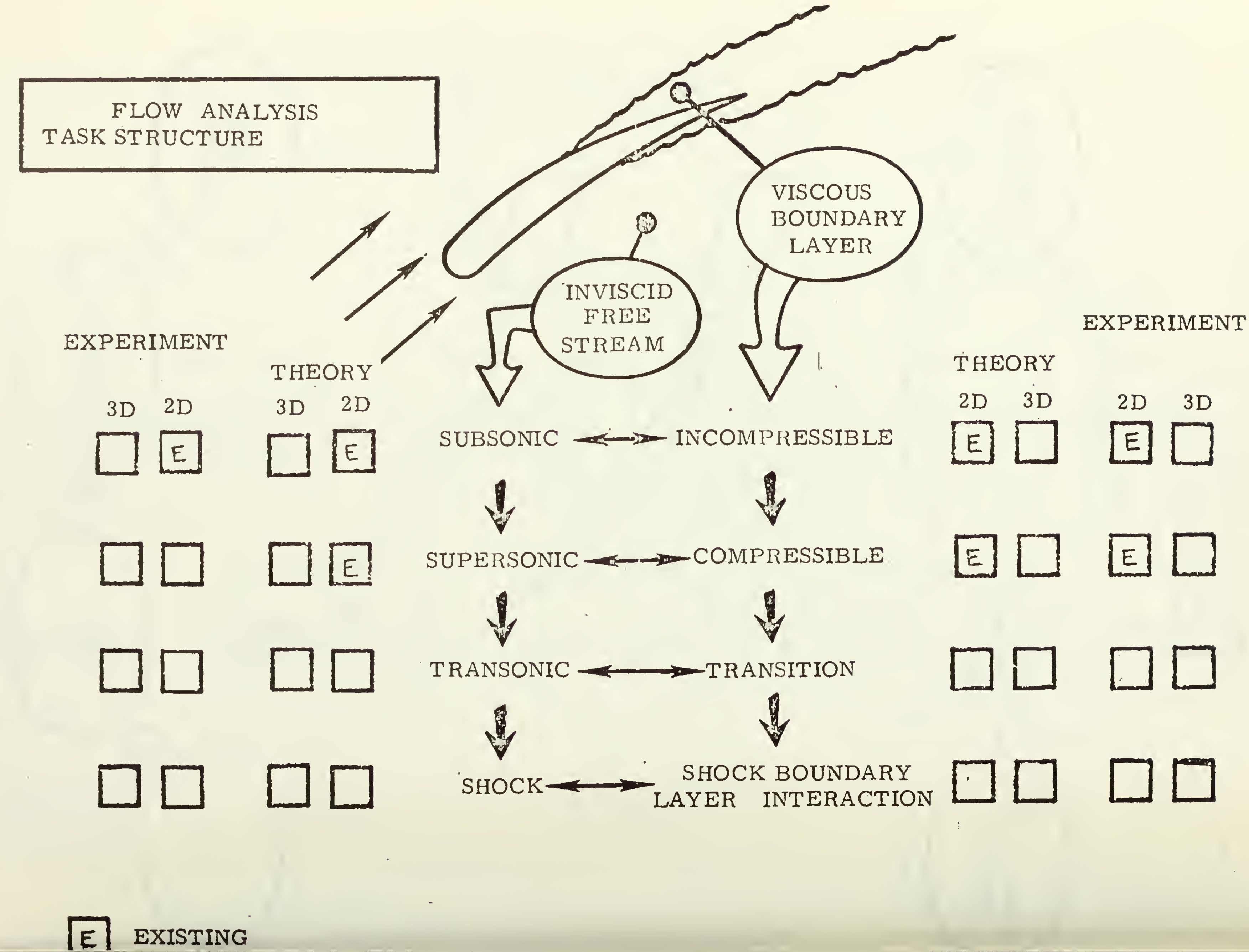
TRAILING VORTICES=SHED VORTICES+TRAILING FILAMENT VORTICES

FIGURE 1



APSI - AIRCRAFT PROPULSION SUBSYSTEM INTEGRATION


TURBOMACHINE
FLOW ANALYSIS
BOUNDARY LAYER
TUNNEL
ANALYTICAL
MODELS
EMPHASIZES
PHYSICAL
PRINCIPLES
CRITICAL
EXPERIMENTS
SUPERSONIC TUNNEL
BOUNDARY LAYER
FREE STREAM
(INVISCID FLOW)
WAKE
(INTERACTIONS)
2D
3D
BLADE ROW
INTERACTIONS
DESIGN
SYNTHESIS


FLOW ANALYSIS
TASK STRUCTURE
INVISCID
FREE
STREAM
VISCOUS
BOUNDARY
LAYER
EXPERIMENT
THEORY
3D 2D
3D 2D
E
E
E
SUBSONIC
INCOMPRESSIBLE
SUPERSONIC
COMPRESSIBLE
TRANSONIC
TRANSITION
SHOCK
SHOCK BOUNDARY
LAYER INTERACTION
THEORY
EXPERIMENT
2D 3D
2D 3D
E
E
E
E
E EXISTING

COMPARISON
OF
"CONVENTIONAL" AND "DESIGN SYNTHESIS" APPROACHES

TEST →	COMPLETE ENGINE	COMPONENT TESTS	LABORATORY EXPERIMENTS
CREDIBILITY →	EXCELLENT	GOOD	LIMITED
COST TIME RESOURCES →	VERY HIGH	HIGH	LOW

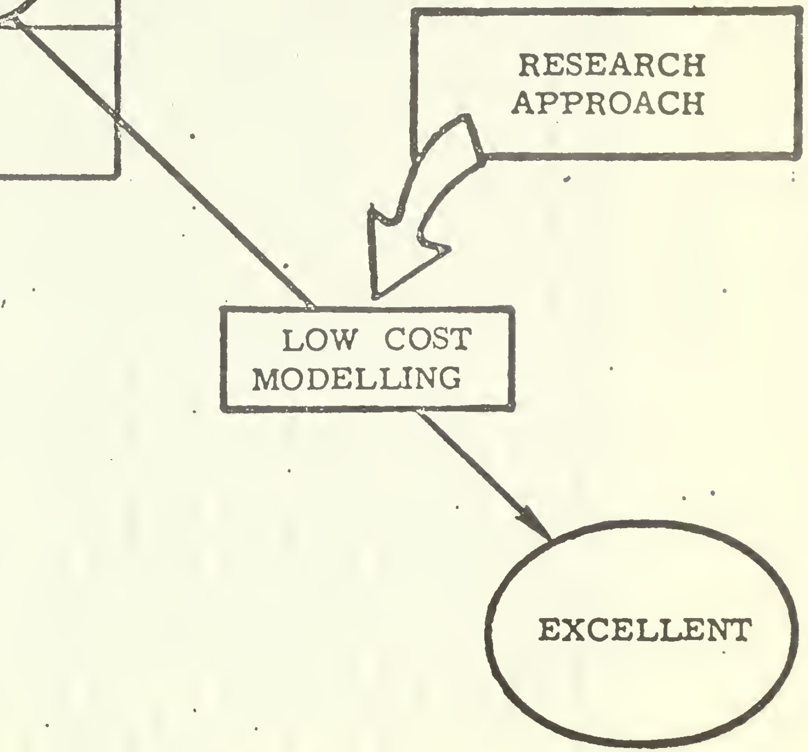


FIGURE 5

DISCUSSION

(Henderson) We had a small non-representative subset of this workshop last night discussing the problem, and that non-representative subset concluded, that it would not occur in our lifetime that we would be applying three-dimensional flow calculations in a straight-forward way to design compressors and other components. But it was concluded that there could be ways in which we would apply some of these newer techniques, to set up a different kind of flow model for calculating components. And I think, although it got later in the evening, it was also concluded by this non-representative subset, that it would be valuable to have some idea as to what it was that we intended to accomplish with the component, in terms of future applications, at the time that we were carrying out the development of the new flow model or design concept. On the basis of that, Wally McBride, one of the participants of this non-representative subset of the workshop, agreed that he would completely revise his presentation, and talk about how you might see future application trends for the applications of these kinds of components. So with that introduction, I will let Wally take over.

(McBride) This is the first time that I have been a non-representative subset. However, we did conclude in our discussion that in industry, compressors are not designed from scratch except very, very rarely. The design problem, which is basically to reach a flow, a pressure ratio, and a high efficiency with a satisfactory operating map, which implies a multiplicity of design points and usually a multiplicity of conflicting requirements, is beyond the scope of our ability to sit down and come up with something that is absolutely right on the money the first time. I am highly encouraged by the fact that we get as close as we do. This is due, of course, entirely to all of the research and all of the analytical solutions that have gone on in the past, and that are continuing into the future. I think in the past day and a half the most encouraging thing I heard was Professor Collins' work on the possibility of improved interstage and overall flow measurement. We desperately need improved flow measurement and test analysis techniques if we are to achieve either or both better machines and design assurance. Today there is too much fiction and opinion in the design-test-analyze-improve design input feedback cycle. I am going to speak at this point largely on what we expect to be coming from the analytical developments, what kind of compressor requirements we are looking forward to in the next few years. The following trend plots are derived from a number of actual machines and advanced component design and system studies. They are unclassified because we have removed the designations which tell which point is the F-101, etc.

Any prediction system has to have some valid framework of measurement if it is to avoid the trap of simple trend extropolation which so often leads to incompatible, if not physically impossible, performance levels because interactions were not properly considered. We cannot accept that we are going to have ever-increasing efficiency, ever-increasing overall pressure ratio. We will talk about that either a little later this afternoon or tomorrow morning. I am going to touch briefly on how these different problems interact upon one another, and go from there. First, let's take a look at some expectations of compressor aerodynamic loading (Figure 1). You will notice that we indicate quite interesting values. This is for loadings at the hub. We are going from values back in the mid '50's of just under unity, and we are predicting up around 1.6 to 1.8 for turboshaft, turbojet engines. The turbofan engines are going not quite as high on loading.

(Unidentified) Is that based on temperature head or turbine head?

(McBride) This is actually enthalpy or temperature head. Figure 2 is the first stage of a fan and the maximum aerodynamic loading. Here at the hub we are running in the vicinity of 1.8 values, and we are not forecasting too much of an improvement, at least not much with non against rotation inlet-guide-vane fans. The first stage at the tip would have a lower value. This is a reflection, of course, of the hub radius, and whether you are going to have uniform or non-uniform inlet to the fan. Another factor comes out of the cycle requirement, because there is no sense in building a fan or compressor which is not useful in an engine. So here (Figure 3) we are looking at expected fan bypass ratios, going from values of today in the five to eight category, going on up perhaps as high as 12:1 for a subsonic, logistics-type aircraft fan and lower ratios for transonic turbofans. This again, as the Admiral said yesterday, depends upon how you define your mission. If you are going to fight, in general it comes out to be a turbojet; but if you are going to get there and fight, it comes out to be a turbofan. Figure 4 is the fan-compressor airflow. This is one guess at how much flow we can cram through the engine. At the front face we are talking of flows probably in the 43 or higher area, but not too much change from where we are today. I personally think this forecast is too low and that there are good reasons, such as distortion tolerance and better over-all engine arrangements, for trying for flow levels of 45-46 lb/sq. ft. of annulus and 40-41 on face area. For the engine core, there is no payoff whatsoever in having high flow per frontal area, so that these levels are lower. We find, in fact, that there is much payoff for not going to high flow, in weight and overall performance. Next, (Figure 5) let's look at some high-speed fans and compressors, and speculate a little bit on tip speed. Tip speeds were running around the 1100 category in 1955, and did not increase too much, until we got into the fan business. We are forecasting that we will probably, with new materials, have levels about 2000 feet per second tip speeds for fans and about 1800 for the turbojet and the core engine. Of course, core engines have to have high physical tip speeds, because of the higher temperatures coming in, in order to get high corrected speeds. Next (Figure 6), we have fan pressure-ratio requirements, again from the cycle. In the mixed-mission military turbofans, we are reaching the "leaky" turbojet by-pass category, and we might see pressure ratios of from about two up to about four. This gets to be a fairly interesting fan if you try to do it in one stage. Single-stage fans are now down in the category of about 1.5, and approaching 1.8. We think that these levels are probably about as high as we care to go. Now the next figure I have is not really a compressor figure, and it is not a fan figure (Figure 7). It does deal with turbines, which, of course, are heated to higher and higher inlet temperatures. This figure shows a typical distribution of cooling air as the inlet temperature goes up. With the turbine inlet temperatures of around 2400 degrees, you can get away with cooling only the turbine nozzle and rotor. However, as you go past that temperature, you begin to have to put some cooling air into your structure. The most interesting thing is that the low-pressure turbine begins to demand as much air for cooling, if not more air, than does your high-temperature turbine. This is for two reasons. The main one is that you have so much more surface. You don't have to cool many degrees in temperature,but you have so much surface that a lot of cooling air becomes necessary. You can work up trends like this for each component in the overall system. The real problem, thinking in terms of this workshop, is how to achieve the results, and this is the introduction from last night's subset discussion. In order to achieve this, we first have to come to some kind of a good analytical, initial design. Then we have to go about developing our engine to achieve what we are committed to do, either as a component or as an overall system. There are many ways of doing this. You can do it in the beginning by thoroughly investigating each element and each component in the system. This would take quite a long time. Or you can go to some type of an experimental approach. Looking at these things very carefully and very quickly we can do an adequate design job in itself. Let's just look at the number of inputs which is involved in this type of approach. I am using 16 variables. (See Table 1) If we want to consider interactions between all 16 variables, we would have 9×10^{10} individual combinations that we have to consider in order to come up with the answer. That takes too long a time to do, so we need another approach.

A next big step in development is to use an adaptive optimization technique, and for 16 variables, that cuts it down to only 1400 combinations. This only takes about 30 years to do, and thus isn't an improvement. By using some experimental design techniques, it is possible to reduce the number of combinations which have to be investigated, either analytically or experimentally, to on the order of about 150 to 120. This becomes barely practical. The best way to get around these difficulties that I know of is to hire a lucky designer who intuitively eliminates all the unnecessary problems and concentrates on the real ones. We can, however, do some things, which I will illustrate, with some of the experimental design techniques. This is taken from an aircraft. We can design and get a mathematical model in the form of a linear-quadratic and first-order interaction model with a relatively few combinations. The experimental design technique says to put together combinations of variables in this order and see what we come up with. Many of these should be trivial solutions in that they should be over the edge of reality and thus establish the limits of the solution. Figure 8 merely illustrates combinations that you get. You could do this with a compressor. We actually started on this approach partially because of the problem of how to optimize a variable-stator compressor. With each stage variable, you have an awful lot of potential compressors. Assuming that we have calculated or tested according to an input table of this nature to find a series of configurations, we can then come up with an analytical model, which we can optimize using a number of different modern mathematical techniques. Here Table 2 is a compressor test plan where we go up to six stages, where we can vary both the rotor and stator stagger, the speed, the discharge throttle position, and the inlet pressure levels. This would give us 120 possible interactions; and if we varied one thing at a time in the elastic plan, it would require about 65,000 tests. With the experimental-design-plan approach, we could do it with 156 tests if we were willing to forego six of these 120 interactions. Normally, we can always structure the program to do this. However, if we can't, it would require 280 tests. One thing about this, frequently some of these test conditions cannot be obtained. But if you have sufficient degrees of freedom in your statistical experiment, it is really a matter of no concern. Once such a plan has been executed either analytically or in a test plan, we've got our analytical model, we go in and use non-linear optimization techniques of which there are a dozen or more quite useful programs available. We come up now with this kind of a problem. We want to get a high pressure ratio, we have a constraint on weight flow and efficiency, we have stress in the various stages, and similarly for all the variables. These are a function of the inlet pressure of the stage, the throttle position, the angle set on rotors and stators, and all the other independent variables. We have to construct an objective or payoff function to say that this is what we want. Let's say that we want to maximize efficiency. But we must achieve this with a weight flow which is equal to or better than some specified value, a pressure ratio which is equal to or better than some specified value, and with at least a prescribed stall margin. We can do this by taking the equation which represents the efficiency as our objective function, and optimizing that subject to the constraints for all these other variables. This works pretty well. The typical problem has a weighting function on efficiency. In this example we have a weighting function on efficiency at 100% speed and also at 62% speed. You must have the efficiency equal to or better than at 100% speed, and also equal to or better at 100% speed with a stall pressure ratio again greater than some value. This type of problem on a computer is quite feasible. It generally runs in about two to five minutes. We think it is quite efficient and that it works pretty well. It is not always used because it is only one approach. Remember it can be done using inputs derived by analytical experiments just as well as it can be done by using test data.

Let us look at a simple example of the optimization process. The function we are to optimize looks like a ship hull turned upside down (Figure 9). It represents a very simple function of this polynomial type of thing that we are speaking about. Here are the steps that we went through to get the optimum, in this case the maximum value of this function. The first step was here, the second step was here, etc. On the tenth step we were within .01% of the optimum. It looks very much like someone took a marble and rolled it down the inside of the hull. However, that is a trivial problem because we have no constraints involved in it. If we put constraints (see Figure 10) on it of this form, blocking out various areas, then we had to start it within a feasible region. So we start here and follow basically the same path. Here in Figure 11 is the result of one particular test on a compressor where we varied the inlet guide vane, the first stator, the second stator, and the third stator. This is the angle of the stator position, and this was running at 100% speed. We then went into the optimization program, and said let us not have any stator that went past about 7.5 degrees or so from its nominal position while maximizing efficiency. What is this efficiency? For each condition, here is the efficiency. Here is the relative flow. You get this trace of efficiency, which is the maximum for that flow. It turns out here that at 100% flow, we get the maximum efficiency possible. This should be true because it is true by definition. These bumps actually are there, and they come about because one stator or another stator reaches a maximum bound, in other words, we are going into another system. We ran this a number of years ago on a J.85, and the first time we ran it, we were very discouraged. The results came out saying that there was nothing significant that you could do to improve this compressor by varying the stators. Anything you varied past the second stage had no effect whatsoever on its performance. This was not in accord with either scientific or engineering judgments and certainly didn't seem reasonable. Also the machine didn't feel right when it was running. We continued to run quite a test program on the machine. However, we eventually gave up on it, never reaching any of the objectives. When we stripped the machine down, we found something interesting. One of the second stators was not picked up in the control system and was floating at random. If we had had more confidence in the procedure at that time we would have said that the machine was not operating like a compressor, and would have stopped the program then and there and started to look for the difficulty. Experiment design provides a good approach to turbomachine testing and also, I think, to design where you have a very complicated system. Do you have any questions?

NUMBER OF SYSTEM DESIGNS REQUIRED

No Variable	Level of Model: 2 Linear	3 Quadratic	5 Interaction	Estimated Adaptive Optimisation	LEADER Expm't Des Model: Box I	Box II	No 1st Interactions
1.							
2.	4	9	25	10	9	9	1
3.	8	27	125	24	15	15	3
4.	16	81	625	64	25	17	6
5.	32	243	3,125	115	27	19	10
6.	64	729	15,625	191	45	29	15
7.	128	2,187	78,125	307	79	47	21
8	256	6,561	3.91×10^5	461	81	49	28
9	512	19,683	1.95×10^6	641	147	83	36
10	1,024	59,049	9.77×10^6	881	149	85	45
11	2,048	2.67×10^5	4.88×10^7	1,152	151	89	55
12	4,096	8.01×10^5	2.44×10^8	1,497	276	148	66
13	8,192	2.40×10^6	7.21×10^8	1,881	278	150	78
14	16,384	7.21×10^6	3.60×10^9	2,353	280	152	91
15	32,768	1.44×10^7	1.80×10^{10}	2,967	282	154	105
16	65,536	4.33×10^7	9.01×10^{10}	4,309	284	156	120

TABLE I.

COMPARISON TEST PLANS

No. Stages	Interactions**	Test Points Required: Classic Plan	Box Plan*	
1	10	43	25	17/6
2	28	273	81	49/15
3	45	1 045	149	85/12
4	66	4 121	276	148/6
5	91	16 413	280	152/30
6	120	65 569	284	156/6

* Variables rotor and stator stagger, speed, throttling, and inlet pressure level
** First interactions, i.e., $x_i x_j$
* Second column lists data points/lost interactions for next higher fractional replication

TABLE 2.

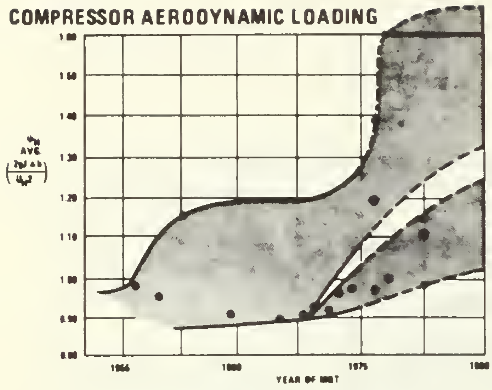


Figure 1.

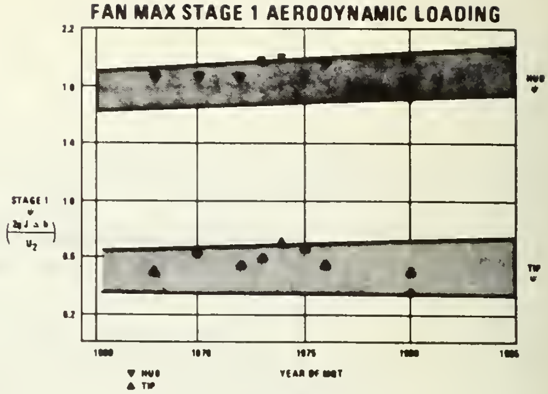


Figure 2.

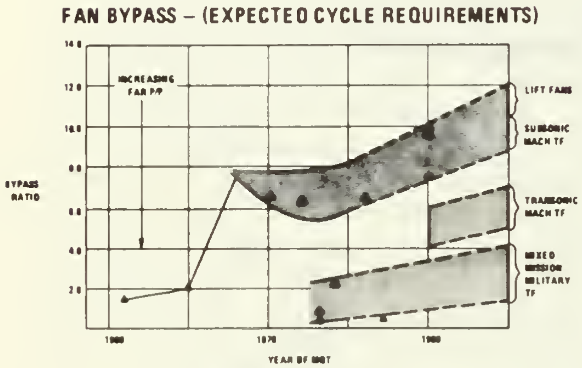


Figure 3.

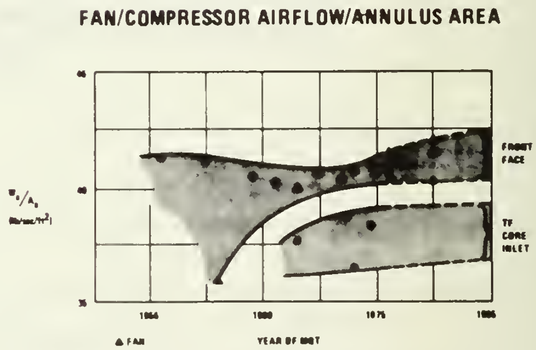


Figure 4.

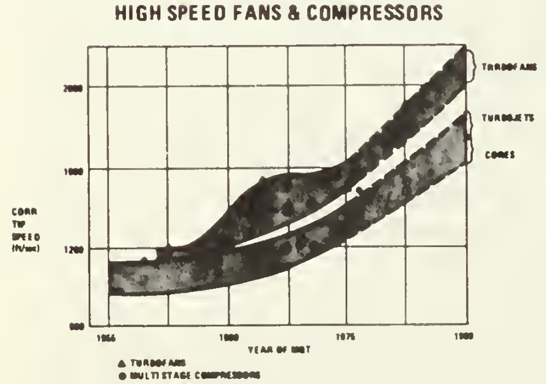


Figure 5.

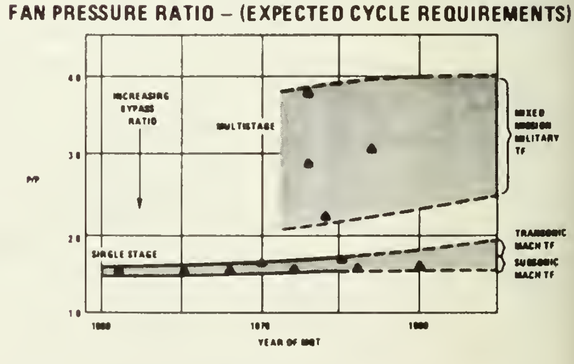


Figure 6.

PICAL DISTRIBUTION OF
C ING AIR REQUIREMENTS
TR PIRATION COOLED TURBINE

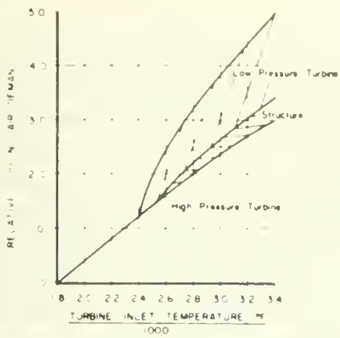


FIG. 7

TYPICAL DESIGN MATRIX

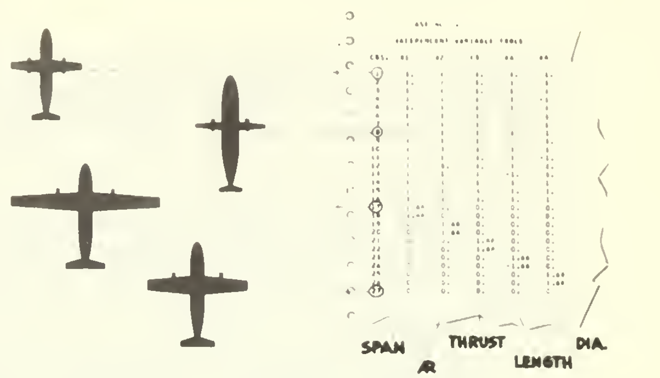


FIG. 8

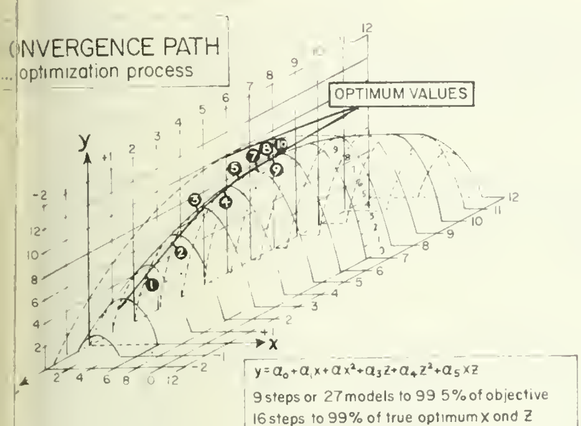


FIG. 9

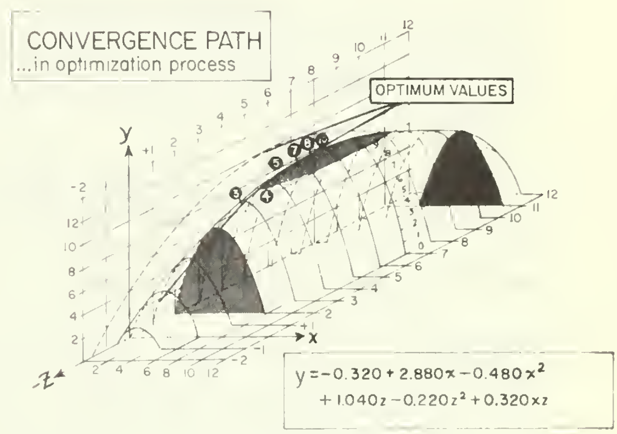


FIG. 10

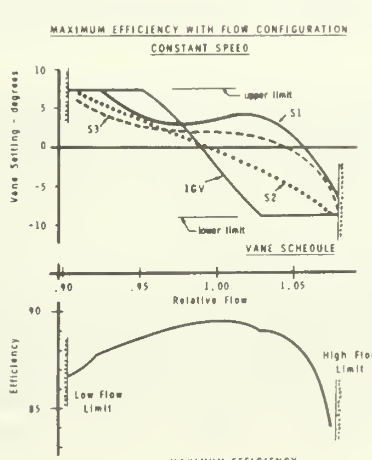


FIG. 11

(Henderson) This kind of approach that you are talking about here, I wonder if I could make a very simplistic statement, so simplistic that the workshop could agree to it. When the fluid dynamicist undertakes a program which is going to contribute to the design of turbomachinery, he should have in mind some kind of a systematic flow model which could employ the results of his research. Is that a reasonable assertion to make? You could fit it in this kind of thing. I think a lot of things follow from that in terms of some of the kinds of fluid-dynamics programs that have been carried out on the basis of applying to component design would not have qualified under that very simplistic kind of statement. Would anybody be willing to comment on that? Or will it be allowed to pass without comment? So the next time a fluid-dynamicist comes up in your company with some kind of a flat-plate kind of boundary-layer system, which he can't relate to a systematic model for design of a component; then you say that is great, he can go ahead and work on that, but don't charge it to any kind of funded dollars related to the component program. I'm not saying it is bad to make some study of something. I'm just saying that there should be some way to relate it to the component.

(Dahlberg) How do you pick those independent variables? I was under the impression that usually you would look at the mission of the aircraft you are interested in, and then take the pressure ratio and the inlet conditions, etc.

(McBride) If you were going to do a mission study, for example, you would pick things like face area of the machine, pressure ratio, and all the major design parameters, efficiency, nozzle areas, etc. If you want a systematic study of those machines, you fly them over the mission. Your objective function then would be to maximize the mission, whatever that might be. A very simple case would be a type of logistics mission, where you would want to minimize the fuel while accomplishing a given range.

(Dahlberg) Well, in this case when you maximize the efficiency, are you sure you are maximizing what you really are after?

(McBride) No, you are not. You have to decide what you want to maximize. I was simply saying that in this example, efficiency was probably what you would choose to maximize. You will notice that one was set up to maximize efficiency weighted between 100% and 60% speed. You can build up a very complicated objective function where you weight everything from pressure ratio on up. However, normally you can impose an engine working limit on here as a constraint, so it must follow along the normal working line. Really, I couldn't care less in the systems problem whether I maximize the efficiency or not. I want the overall performance of the machine. Frequently the important parameters just fall out of the solution. Efficiency is generally very important; size, weight, and cost, which can be involved, can also be brought into the solution.

(Dahlberg) Did you make any progress in the direction of the closed-form approach, calculus of variations approach, or is it all search techniques?

(McBride) Basically, we generally ran into trouble every time we tried the variations approach. We were unable to use them. I don't know whether this was because I'm a lousy mathematician or just what, but they were not successful.

(Huffman) The restraining conditions usually limit the type of approach. In an unconstrained problem, you can use calculus of variations, but when you start talking about finite constraints, then you are in the search domain. In either case, you end up with a numerical method.

(McBride) We spoke previously about component performance trends and the difficulty of getting meaningful measurement parameters which are adequately free of interaction effects. The following figures representing some of our efforts in these directions may be of interest. I am not going through the processes used to adjust each trend curve to a consistent configuration since these methods are properly a topic for a session in itself but are really not important here.

Figure 1. Aerodynamic load index here is a simple statistic plot of stage useful work or pressure coefficient (unity at zero flow) to tip Mach number. It shows that achievable work coefficient and efficiency both fall off with increased Mach number. Thus, the common assumption that inversed tip speed gives powerfully inversed boost pressure is only partially correct, the increase being more nearly linear than quadratic with speed. In addition, the estimated limit line shown is for a relative high hub ratio machine. Lower hub ratios will have lower limiting values corresponding to approximately constant hub loadings.

Figure 2 supplements the previous slide by giving the effect of fan pitch speed adjusted to constant flow rate, radius ratio, work coefficient, etc.

Figure 3 also supplements the first two figures by showing the trend of efficiency with pitch speed for compressors. This figure is also adjusted to a compressor of fixed flow rate, fixed inlet radius and fixed specific flow rate. Flow coefficient thus decreases with increased pitch speed.

Figure 4 shows an estimate for the effort of specific flow rate in the annulus adjusted to a fixed stage configuration. It suggests that above 42-43 lb/sec/sq ft efficiency begins to drop off fairly drastically. This is true for the design procedures used. The big questions are:

- o are other and better design approaches possible?
- o is there any payoff in high flow rates above, say 40 lb/sec/sq ft?

Figure 5 shows the affect of work coefficient on fan efficiency. My reaction to the scatter in this curve is that selection of work coefficient level is not very important in comparison to how well the design is executed.

Figure 6 presents the effect on efficiency of inlet radius ratio. Here the adjustment to a specific set of conditions suggests that perhaps hub loading and Mach number need to be factored in more explicity and that the knee of the efficiency drop might shift dramaticaly if other normalizing parameters had been adopted, e.g. higher tip speed and flow rate. Again the question is what are the conditions for a maximum of a multi-dimensional set?

Figure 7 gives a condition of size on efficiency. It should be noted that this is properly a size and not a Reynolds number condition. Thus, as size increases tip and leakage clearances and manufacturing inaccuracies become less important. This is not necessarily true with increased Reynolds number.

Figure 8 gives a guess for the effect of changing materials on compressor rotor weights. It does not give any consideration to possible aero-mechanical instability boundary shifts with density and modulus.

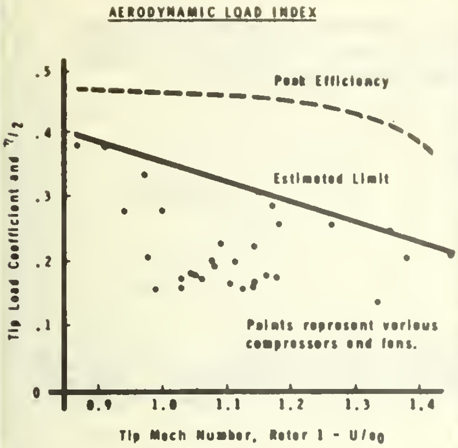


Figure 1.

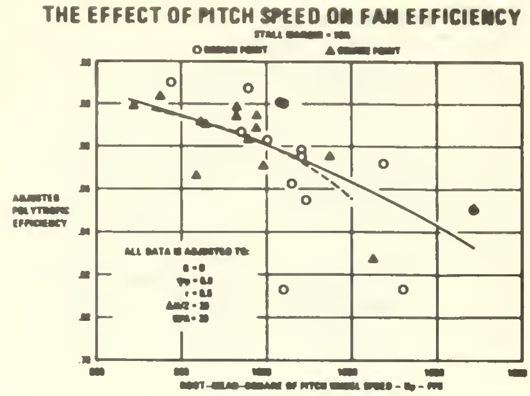


Figure 2.

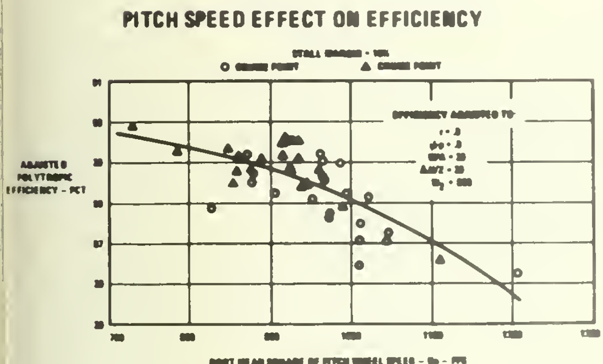


Figure 3.

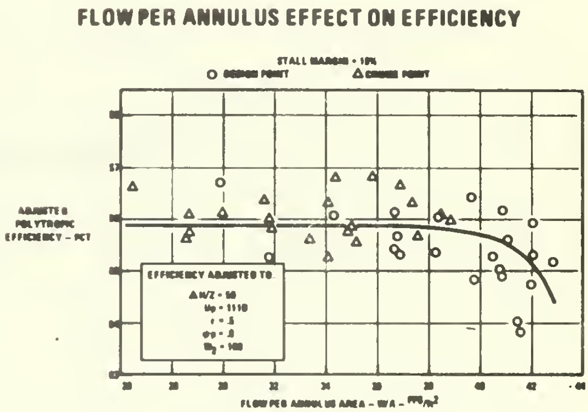


Figure 4.

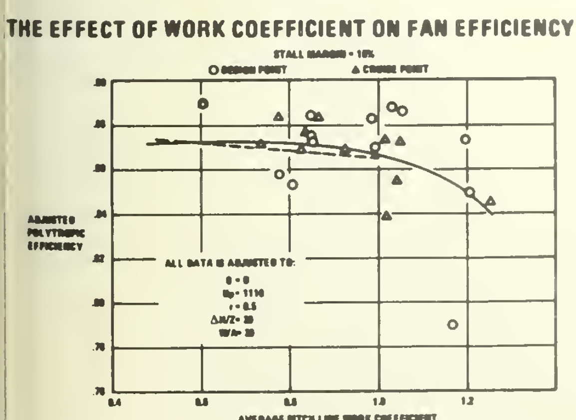


Figure 5.

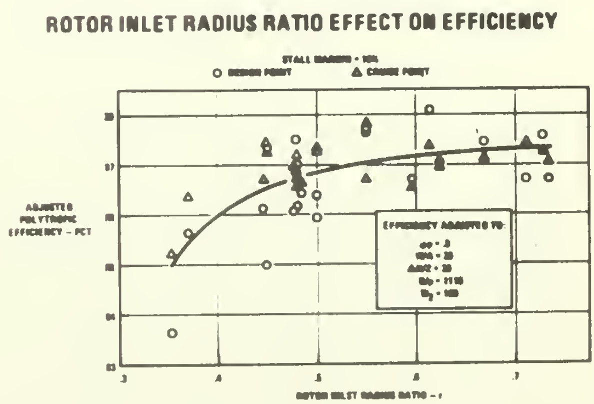


Figure 6.

SIZE EFFECT ON COMPRESSOR EFFICIENCY

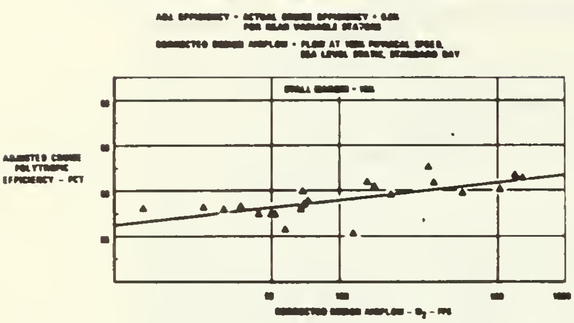


Figure 7.

THE EFFECT OF MATERIALS AND SPECIAL OESIGN ANO MANUFACTURING TECHNIQUES ON FAN ANO COMPRESSOR ROTOR WEIGHT

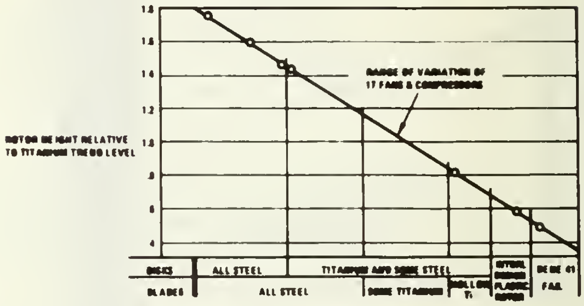


Figure 8.

LOSS EVALUATION METHODS

Discussion Leader: Prof. B. Lakshminarayana

LOSS EVALUATION METHODS IN AXIAL-FLOW COMPRESSORS*

by

B. Lakshminarayana

*Editor's Comment: This is the text on which Professor Lakshminarayana based his presentation.

LIST OF SYMBOLS

R_e	= Reynolds number
T	= Turbulence intensity (inlet)
M	= Mach number
t	= Blade thickness
σ	= Blade solidity
r	= Radius
ε	= Flow turning
δ	= Boundary layer thickness
ω	= Frequency
U	= Flow velocity
$\frac{\omega\delta}{2\pi U}$	= Free stream unsteadiness parameter (Ref. 3)
ϕ	= Flow coefficient
λ	= Clearance/chord ratio
dp/dx	= Local pressure gradient (axial)
dp/dr	= Local pressure gradient (radial)
$\frac{2\Omega\delta}{U}$	= Rotation parameter (Ref. 14)
A	= Aspect ratio of the blade
H	= Shape factor
θ	= Momentum thickness
C	= Chord length
L	= Total length of the compressor (multi-stage)
β	= Flow angle
τ	= Tip clearance
Γ	= Blade circulation
φ	= Sweep angle

h = Blade height

ψ = Blade loading coefficient = $\frac{2C_p(\Delta T_o)}{\rho U^2}$

η = Efficiency

$\Delta\eta$ = Change in efficiency

ζ = Stagnation pressure loss coefficient $\left(\frac{P_1 - P_2}{\frac{1}{2}\rho U_1^2}\right)$

$\bar{\zeta}$ = Passage averaged loss coefficient

P = Stagnation pressure

T_o = Stagnation temperature

C_{D_i} = Induced drag coefficient of the blade

$C_{D_{is}}$ = Induced drag coefficient due to secondary flow

$C_{D_{ic}}$ = Induced drag coefficient due to clearance flow

$C_{D_{it}}$ = Induced drag coefficient due to mainstream secondary flow

D_z = Total kinetic energy in secondary flow

L/D = Lift/Drag ratio of the blade

C_p = Static pressure rise coefficient of the cascade

Subscripts

t = trailing edge

r = radial

S = streamwise

m = meridional

A = annulus or hub wall

B = blade

1 = inlet

2 = outlet

c = due to clearance

i = inner radius

o = outer radius

a = axial

LOSS EVALUATION METHODS IN AXIAL FLOW COMPRESSORS

The purpose of this paper is to classify the sources of losses, the flow and blade parameters on which they depend and then indicate methods of evaluating these losses. An attempt is also made to indicate areas where additional research is needed.

Considerable progress has been made over the last two decades in understanding and predicting flow departure and losses in axial flow compressors. An attempt has been made in Table 1 to bring them together, classify them and indicate the parameters on which they depend.

I. PROFILE BOUNDARY LAYER AND WAKE MIXING LOSSES:

a. Two-dimensional:

The two-dimensional profile and mixing losses depend on Reynolds number, Mach number, longitudinal curvature of the blade, inlet turbulence, free stream unsteadiness and the consequent unsteady boundary layers, main stream pressure gradient, shock strength; blade parameters such as thickness, camber, solidity, sweep, skewness of the blade, stagger angle, gap width (in the case of tandem rows) and blade roughness.

Considerable progress has been made in predicting the 2D profile losses. Papailiou in an earlier paper in this workshop has given an excellent summary of the boundary layer calculation methods. Even though Speidel and Scholz's[2] method is widely used at the present time, an extension of Bradshaw's[4,5] turbulent field methods to cascades should lead to a more accurate assessment of the profile losses under all operating conditions.

The effect of inlet turbulence and Reynolds number on profile losses has been investigated in References 9, 10, 11 and 28. Figure 1, reproduced from Ref. 10, indicates appreciable effects of Reynolds number and turbulence on the rotor performance. Shaw[10] observed decreasing pressure rise with decrease in Reynolds number. It can also be seen from Figure 1 that the static pressure rise decreases when there is inlet turbulence and IGV wakes. The results of Schlichting and Das[8] indicate that the inlet turbulence has appreciable effect on profile losses only for intensities greater than 3%. For $T < 3\%$, the turbulence has favorable effect on boundary layer, whereas for $T > 3\%$, energy lost is increased by increased dissipation. Evans[28] concluded that the boundary layer transition and separation are found to be sensitive to the influence of free stream turbulence at low Reynolds number. At higher Reynolds numbers, the effect of turbulence on boundary layer growth and losses are not of any serious consequence to justify systematic research.

Influence of Mach number and blade thickness on profile losses has been investigated by Schlichting and Das (Ref. 9). The influence of blade thickness on profile losses is found to be significant at low Reynolds number ($\sim 10^5$), whereas at higher Reynolds number (4.10^5), the increase in thickness had little effect on profile losses. The compressibility effects are large only for $M_{a_1} > 0.6$ (See Fig. 2), beyond which the losses are found to increase rapidly. The effect of compressibility is found to be larger for cascades with thicker profiles at low Reynolds number. This is clear from Fig. (2), which is reproduced from Ref. 9.

Table 1 in Papailiou's paper summarizes the overall influence of special effects such as longitudinal curvature, blade sweepback, low Reynolds number, inlet turbulence and unsteadiness. The effect of sweep on losses is investigated in detail by Stark.[27] At high subsonic Mach numbers, the losses decrease rapidly with sweep angle. But for $M < 0.7$, the sweep has negligible effect. In fact, sweep increases the losses for $M < 0.7$, as can be seen in Fig. 3. Very little research has been reported on the effects of curvature and free stream unsteadiness on boundary layer growth along the blades.

The flow through tandem cascades has been investigated by Railly et. al.[12]. Fig. 4, reproduced from Railly's paper, shows the performance of a tandem row. Comparison of C_p and L/D ratio of tandem blade cascade with and without end wall suction is shown for gap chord ratios of 0 and 0.25 inches. Significant improvement in stalling performance is observed in the case of tandem rows. Whether such an improvement can be attained in a real environment of a compressor is not yet established. In fact, recent results from Pratt and Whitney Aircraft Company[29] indicate increased losses in a slotted stator. The flow behavior and viscous losses are very sensitive to the geometry and location of the gap in the case of a tandem rotor and geometry and location of the slot in the case of a slotted rotor or stator. Additional theoretical and experimental investigation of viscid flow through such tandem and slotted rotor or stator is needed to prove its feasibility in aircraft engines.

The prediction of shock losses has been carried out by many investigators; none of them has been consistantly successful for predicting the losses at conditions other than design. The earlier prediction method is due to a group at NASA Lewis Research Center

(Ref. 30). These authors consider a normal shock wave in the entrance region and use a Prandtl-Meyer expansion along the suction surface of the blade to obtain the suction surface Mach number at the assumed intersection of the shock wave with the suction surface. The pressure recovery due to normal shock is then used to calculate the pressure losses. Balzer[13] later improved this method by using the relative fluid turning angle in place of suction surface camber for predicting the shock losses. In addition he uses two-dimensional channel flow rather than Prandtl-Meyer turning for estimating the Mach number at which shock occurs. The results of both prediction methods are compared with experimental results in Fig. 5, reproduced from Balzer's paper. Balzer's method seems to agree with experimental results better.

b. Three-dimensional:

Profile and wake mixing losses depend on (in addition to those listed in Ia); rotation parameter $\frac{2\Omega L}{U}$ or $\frac{2\Omega\delta}{\bar{U}}$, blade twist, radial pressure gradient, the aspect ratio of the blade and the meridional flow curvature in the mainstream region. Johnston[14] and Horlock[31] have given a summary of the physical nature of 3D blade boundary layers and approximate methods of predicting these. But the magnitude of losses, the shear stress distribution and the effect of cross or radial flow on transition and flow separation should be systematically investigated.

The losses resulting from the mixing of three-dimensional wakes depend on the nature and magnitude of three-dimensional velocities, the momentum thicknesses and the turbulence intensities. Unfortunately, only a two-dimensionl treatment of the wake is available for predicting the wake mixing losses. A two-dimensional model is oversimplified because operational rotors generate the radial component as well as other components, and furthermore, the wake operates in a centrifugal flow field. Hence, a three-dimensional treatment of the wake diffusion and the associated mixing losses is needed. Method of measuring the mean properties of the 3D wake is reported in References 7 and 24.

II. ANNULUS AND HUB WALL BOUNDARY LAYERS:

I call this as the direct effect, that is, I neglect the interaction between the blade boundary layer and the annulus wall boundary layer, leakage and secondary flows occuring in this region. Methods of computing the annulus wall boundary layer and their effect on performance is proposed by Horlock,[31] Mellor,[17] Smith[16] and Dettermering and Kensenhoff.[32] The last paper deals with computation of entropy and enthalpy gradients through the end wall through the introduction of shear stresses in the equations of motion. They were thus able to compute the direct skin friction loss, work done factor and the flow blockage factor. Mellor[17] introduces new concepts such as axial and tangential force defect thickness near the end wall. In deriving the theory he identifies the role of secondary flow and tip clearance flows; hence, his calculation method includes some of the losses listed under item 3 in Table 1. Mellor's theory identifies all the physical quantities that govern the flow and provides a method of calculating the losses due to end wall layers. Figure 6, where Mellor's calculation of stage characteristics of two hypothetical compressors is shown, indicates the most remarkable aspect of the theory in as much as the stall is predicted.

III. SECONDARY FLOW

a. Corner Stall:

The losses due to corner stall occuring at the intersection of end wall and the suction surface of the blade depends on flow turning, Reynolds and Mach numbers, momentum thickness and shape factor of wall boundary layer at inlet, blade boundary layer, inlet turbulence and unsteadiness. It should also depend on rotation factor $(\frac{\Omega\delta}{U})$, since the blade boundary layer is thrown outwards in the case of rotor. Hence, the corner separation near the hub is less severe in the case of a rotor as compared to stator.

An attempt was made by the author[19] to systematically study the various effects near the end wall. In Fig. 7, the spanwise variation of loss coefficients measured at various inlet conditions are plotted. Leakage flow only was studied by means of a slot at midspan of a cascade. Whereas, the corner stall was studied with a wall at the midspan for $\lambda = 0$ and 0.04. The plot (Fig. 7) indicates that the corner stall dominates even in the presence of leakage flow. A semi-theoretical analysis of corner stall was carried out by Hanley[18] whose correlations are shown compared with his experiment in Fig. 8. Using Cole's profile for the end wall boundary layer (without skew), Hanley was able to correlate the losses. The end wall loss correlation suggests that the corner stall is primarily a function of pressure rise and inlet boundary layer parameters. The skewness in the inlet boundary layer profile, which exists in an actual environment, is likely to give rise to losses which are substantially different from those suggested by Hanley. Hence, further investigation of losses in an axial compressor stage (where inlet velocity profiles are skewed) is needed.

A theoretical analysis of the corner flow in the absence of pressure gradient and flow turning is carried out by Perkins[26] and Gersten[25]. This flow model is much too idealistic for its application to axial flow compressor. The boundary layer near the wall is highly three-dimensional and it operates in adverse pressure gradient. Any meaningful theory should incorporate all these effects.

b. <u>End Wall Secondary Flow</u>:

The secondary flow losses (in the absence of corner stall) depend on flow turning, Reynolds and Mach number, rotation parameter $(\frac{\Omega\delta}{U})$, shape and momentum thickness of inlet velocity profile, blade solidity, loading of the blade and free stream unsteadiness $(\frac{\omega \delta}{2\pi U})$. The classical work on this flow is due to Hawthorne[22], and the author later improved the predictions by allowing for Bernoulli surface rotation and for change in boundary layer characteristics through the cascade. The kinetic energy in secondary flow, which gives qualitative information of the losses due to secondary flow, derived from Hawthorne's[22] and author's[20] predictions are compared with Soderberg's[33] data in Fig. 9. It is evident that it is essential to include some of the nonlinear effects if an accurate prediction of the losses is sought. The effect of centrifugal force field and blade rotation on secondary flow needs further investigation.

c. <u>Mainstream Secondary Flow</u>:

Mainstream secondary flow arises due to trailing vortices caused by radial variation in circulation. The dominant parameters in this case would be gradient in blade circulation and blade solidity. Various methods available for predicting the loss due to this secondary flow are reviewed in Reference 21.

d. Tip Clearance Flow:

The tip clearance losses depend on lift coefficient, tip clearance height, solidity, inlet boundary layer shape and thickness Mach number, Reynolds number, flow coefficient, aspect ratio and shape of the blade tip (References 15, 19, 21, 23). The drag coefficient due to leakage vortex predicted by the author using a simplified model (two rows of infinite vortices spaced at twice the clearance height) is shown compared with experiment in Fig. 11. The agreement is quite good. Later (Ref. 23), the author used this model to derive a correlation for loss in efficiency due to clearance. This calculation allows for spanwise velocity inside the blade boundary layer at the tip. This correlation agrees well with the measured values as shown in Fig. 12.

The latest flow model[23] proposed for accurate evaluation of tip clearance losses is based on the fact that the vortex in realt has a viscid core surrounded by a potential or 'free vortex'. Varics contributions to tip clearance losses derived from this model are shown in Fig. 13. This model accurately predicts the presence of peak losses, observed away from the blade tip. This model is thus able to predict the spanwise variation in losses which is one of the most important information needed for the three-dimensional flow analysis using complete equations of motion. The predictions agree well with values measured in author's cascade (Ref. 9).

The presence of 'scraping vortex' observed by Allen and Kofskey[34] remains unexplained. No detailed investigation of the losses associated with this vortex has been made. It is likely to depend on (tip clearance/annulus wall boundary layer thickness) rati, blade loading and flow coefficient.

IV. INTERACTION BETWEEN THREE-DIMENSIONAL BLADE BOUNDARY LAYER AND ANNULUS WALL BOUNDARY LAYER.

The radial flow inside the blade boundary layer at the tip interacts with the annulus wall layer to produce a complex flow phenomena. Very little is known about the energy loss associated with this mixing. The present practice is to budget this loss under the 'end wall losses'. Recent investigations at Penn State on a single rotating helical blade revealed that the interaction results in an increase in boundary layer thickness towards the tip.[40] This can be qualitatively explained by the fact that the radial velocities decrease toward the tip, thus accumulating all the low energy fluid towards the tip. Considerable experimental and theoretical investigation is needed before a loss model can be proposed for this effect.

V. WAKE CHOPPING EFFECT

A simple model has been proposed by Kerrebrook and Mikolajczak[35] for the transport of energy by the rotor wakes passing through stator. Energy transport inside the stator leads to impingement of the wake flow on the stator surfaces and may give rise to secondary flow inside the blade passage. This phenomena is likely to prevail in the entire passage from hub to tip. The loss associated with this wake transport is found to be small compared to two-dimensional profile losses. But it may become appreciable in rotors with large solidity.

FUTURE RESEARCH

Future research should be directed in understanding

(a) the loss associated with the three-dimensional boundary layer on the rotating blade and the resulting 3D wakes. The major effect of rotation is to produce large gradient in loss towards the blade tip.

(b) the losses associated with the free stream turbulence and free stream unsteadiness such as caused by inlet distortion and upstream wakes.

(c) the loss associated with the interaction between 3D blade boundary layer and annulus wall boundary layer.

(d) the losses associated with the corner stall in the presence of skewed inlet boundary layer.

(e) the losses associated with the passage shocks and the wake transport.

REFERENCES

1. H. Schlichting and Scholz: Ing. Archiv 29, (1951), 42-65.

2. L. Speidel & N. Scholz: VDI Forschung 464 (1957)

3. P. Bradshaw: Proc. Int. Symposium on Fluid Mechanics and Design of Turbomachinery, Aug. 31-Sep. 3, 1970, Penn. State (To be published as NASA SP)

4. P. Bradshaw. et al: J. Fluid Mech. Vol. 28, pp. 593-616 (1967)

5. P. Bradshaw: N.P.L. Aero Rep. 1286, Also British ARC 30910 (1969)

6. P. Bradshaw: J. Fluid Mech. 36 177 (1969)

7. A. Poncet & B. Lakshminarayana: NASA CR 72750 (June 1970)

8. H. Schlichting & A. Das.: Flow Research on Blading, L. S. Dzung (Ed), (1970)

9. H. Schlichting & A. Das.: J. Basic Eng, March 1966

10. R. Shaw: Proc. 1 Mech. E, (London), Vol 184 (1969-70)

11. S. A. Dovzhik: Wright Patterson AFB FTD-TT-62 17750 (Translated from Russian)

12. J. W. Railly et. al.: Proc. 1 Mech. E., Vol 184 (1969-70)

13. R. L. Balzer: ASME Paper No. 70 GT 57

14. J. P. Johnston: Proc. Int. Symp. Fluid Mech. & Turbomachinery, 1970.

15. G. Hubert & K. J. Bauermeister, VDJ Forchungsheft 496

16. L. H. Smith, Jr.: Flow Research on Blading, L. S. Dzung (ed), 1970

17. G. L. Mellor: ASME Paper 70 GT-80

18. W. T. Hanley: ASME Paper 68 GT-15

19. B. Lakshminarayana: British ARC R & M 3483

20. B. Lakshminarayana: J. Basic Eng., March 1967

21. B. Lakshminarayana: Int. J. Mech. Sci., 1963

22. W. R. Hawthorne: British ARC Report 17, 519, 1955

23. B. Lakshminarayana, J. Basic Eng. Sept. 1970

24. G. Muesmann: Z. Flugwiss, Dec. 1958

25. K. Gersten: AGARDO GRAPH 97, May 65

26. H. J. Perkins: J. of Fluid Mech., Vol 44, Part 4, pp 421, 1970

27. U. Stark: DLR Forschung 67-09 (1967)

28. B. J. Evans. Cambridge University Report CUED/A-Turbo/ T.R.23, Sept. 1970

29. Pratt & Whitney Report PWA FR 2286, NASA CR 54 549, 1967

30. Schwenk, F. C., Lewis, G. W., and Hartmann, M. J., NACA RM E57 A30, 1957

31. J. H. Horlock, Flow Research on Blading, L. S. Dzung (ed).

32. W. Dettermering and J. Kensenhoff. ASME Paper 68 FE 54.

33. C. R. Soderberg: Gas Turbine Lab. Report 46, M.I.T. (1958)

34. H. W. Allen and M. G. Kofskey, NACA TN 3519 (1955)

35. J. L. Kerrebrock and A. A. Mikolajczak: ASME Paper 70 GT 39

36. K. Fujie: Bull Japan Soc. Mech. Eng. Vol 5, No. 18, 1962

37. M. H. Vavra: Aerothermodynamic and Flow in Turbomachines, P. 381.

38. J. L. Jefferson and R. C. Turner: International Ship Building Res. Progress. Vol 17, p. 20, 1960.

39. A. D. Williams: A. E. Thesis, California Inst. Technology, 1960.

40. A. Jabbari: M. S. Thesis, Aerospace Engineering, The Penna. State University, 1969.

TABLE I

Classification of Losses in Axial Flow Compressor

I. Profile Boundary Layer and Wake Mixing Losses

Two-dimensional: (R_e, T, M, $\frac{\omega\,\delta}{2\pi U}$, t, σ, r_t, ε, ϕ, gap width (tandem), blade roughness, dp/dx, surface curvature, φ)

Three-dimensional: (In addition to those listed under 2D: $\frac{dp}{dr}$, $\frac{2\Omega\delta}{U}$, A, H_R, H_S, θ_R, θ_S, T_S, T_R, C, r_m)

II. Annulus and Hub Wall Boundary Layers (Direct)

(R_e, M, $\frac{\omega\,\delta}{2\pi U}$, T, dp/dx, curvature, H_S, θ_S ε, L, ϕ, σ)

III. Secondary Flow

Corner Stall: (ε, R_e, M, δ_A, δ_B, T, $\frac{\omega\,\delta}{2\pi U}$, $\frac{\Omega\delta}{U}$, H_A, H_B, Inlet skewing)

End Wall Secondary Flow:

(ε, R_e, M, δ_A, δ_B, T, $\frac{\omega\,\delta}{2\pi U}$, $\frac{\Omega\delta}{U}$, H_A, H_B, β_1, C_L, σ, A)

Tip Clearance Flow:

(C_L, τ , δ , R_e, M, σ, β_1, β_2, δ_A, δ_B, H_A, H_B, t, tip shape, ϕ, A)

Main Stream Secondary Flow:

$$(R_e,\ M,\ d\Gamma/dr,\ \sigma,\ A,\ \frac{\omega\ \delta}{2\pi U})$$

Scraping Effect: $(\phi,\ \delta_A/\tau,\ C_L)$

IV. Interaction Between 3D Blade Boundary Layer and Annulus Wall Boundary Layer at the Tip

V. Wake Chopping Effect

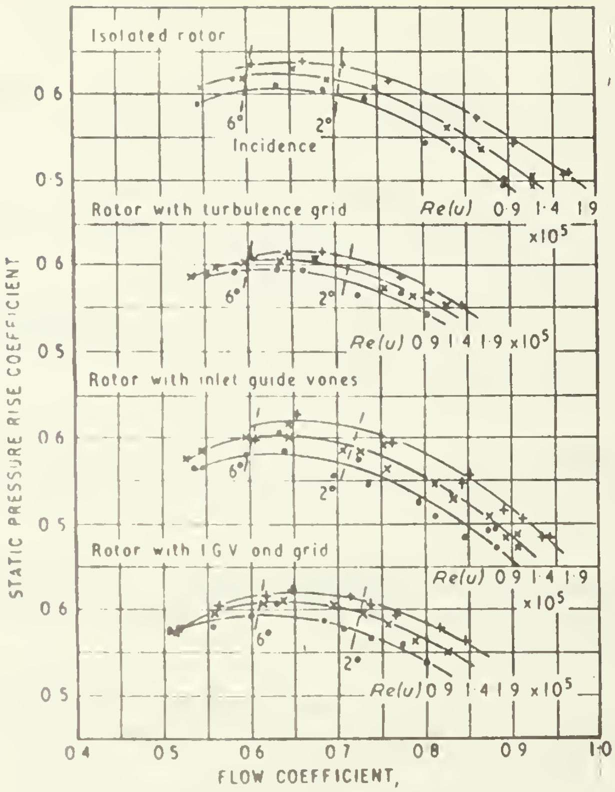


FIGURE 1: VARIATION OF COMPRESSOR PERFORMANCE WITH REYNOLDS NUMBER AND TURBULENCE INTENSITY
SHAW (REF. 10)

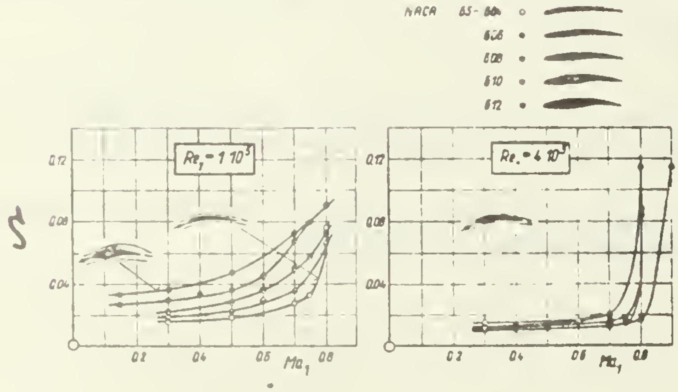


FIGURE 2: INFLUENCE OF MACH NUMBER ON LOSS COEFFICIENTS OF COMPRESSOR CASCADE WITH DIFFERENT BLADE THICKNESS $\sigma = 1.0$, $\beta_1 = 50°$

SCHLICHTING AND DAS (REF...

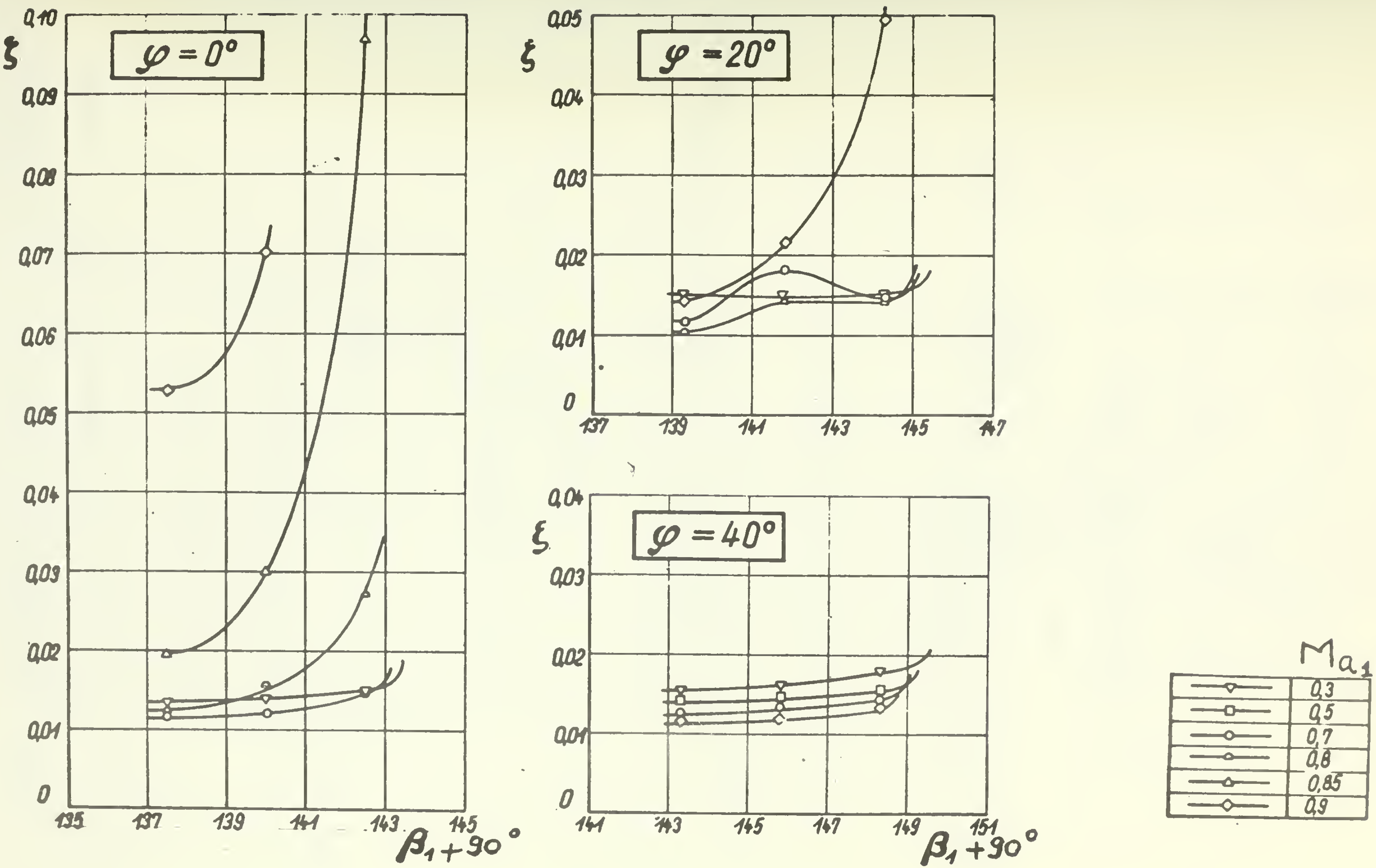


FIGURE 3: INFLUENCE OF SWEEP ANGLE (φ) ON LOSS COEFFICIENT IN A CASCADE AT DIFFERENT MACH NUMBERS ($\sigma \simeq 1.0$, NACA 65-608 PROFILES, $R_e = 4.10^5$)
U. STARK (REF. 27)

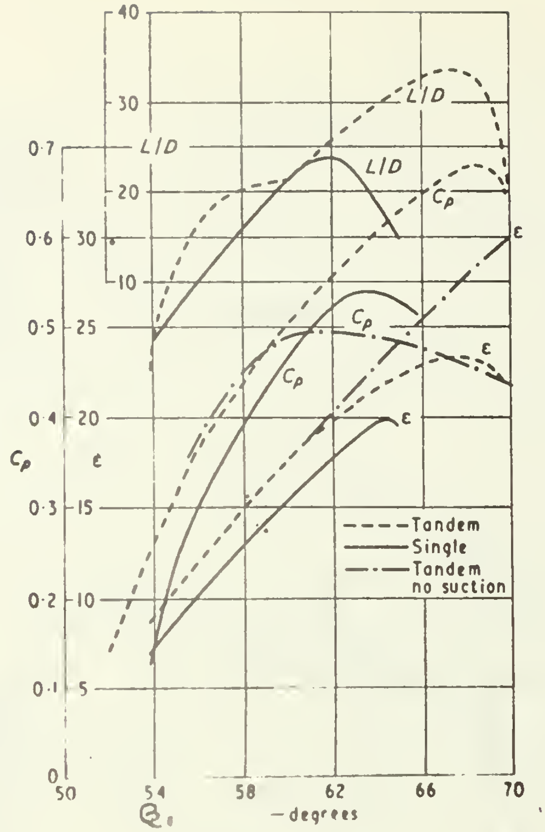


FIGURE 4: PERFORMANCE OF TANDEM CASCADES [σ = 0.94, STAGGER 55° (FIRST BLADE) 40° (SECOND BLADE), GAP WIDTH 0.25 INCHES]
RAILLY ET AL (REF. 12)

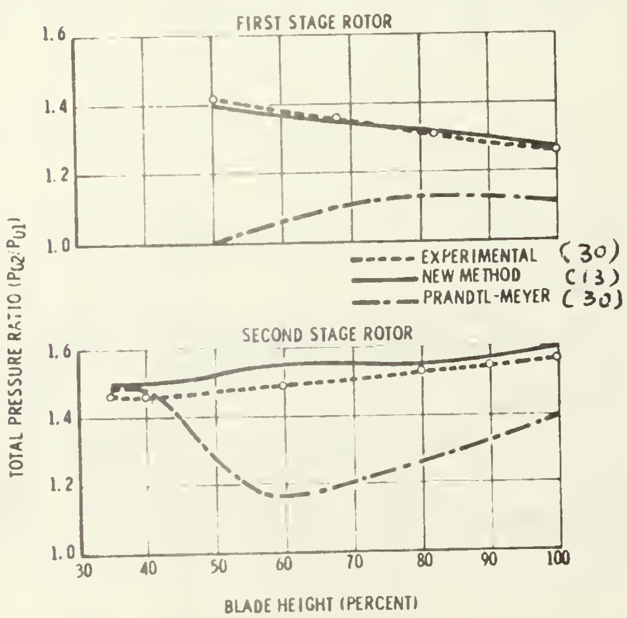


FIGURE 5: EXPERIMENTAL AND PREDICTED RADIAL TOTAL PRESSURE RATIO DISTRIBUTION FOR THE NACA FIVE STAGE TRANSONIC COMPRESSOR

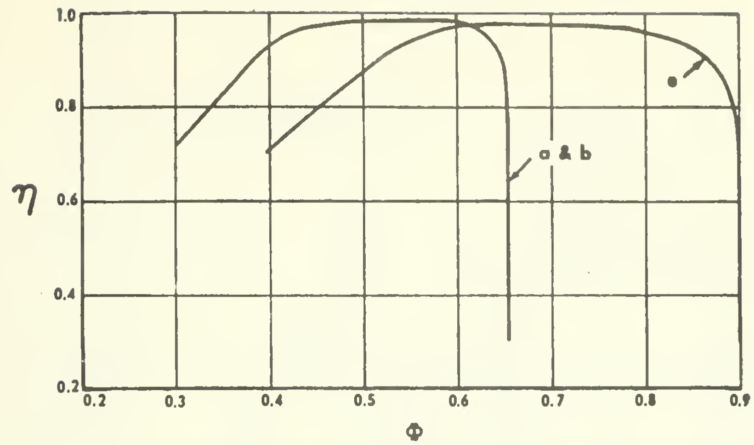


FIGURE 6: STAGE CHARACTERISTICS OF A SIX STAGE HYPOTHETICAL COMPRESSOR ALLOWING FOR END WALL BOUNDARY LAYER GROWTH
MELLOR (REF. 17)

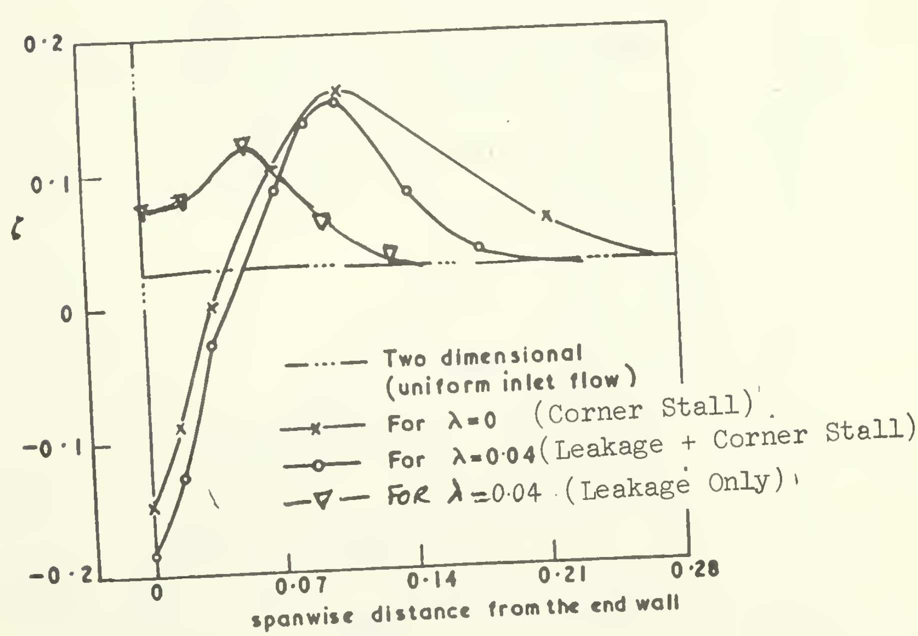


FIGURE 7: SPANWISE VARIATION OF LOSS COEFFICIENT DUE TO CORNER STALL, LEAKAGE AND COMBINED EFFECTS IN A CASCADE
LAKSHMINARAYANA (REF. 19)

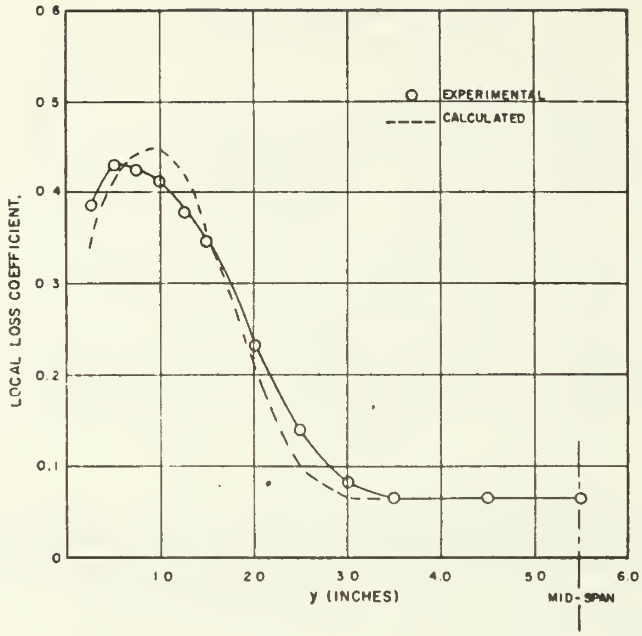


FIGURE 8: EXPERIMENTAL AND CALCULATED LOSS VARIATION IN END WALL REGION OF A CASCADE
HANLEY (REF. 18)

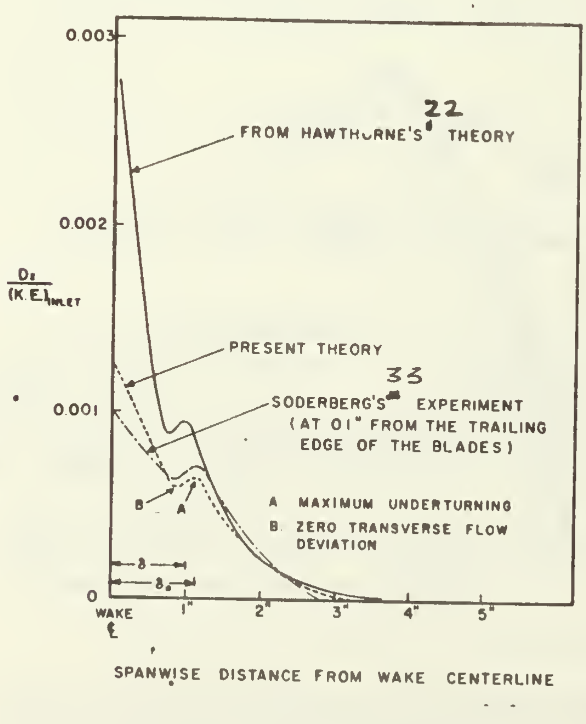


FIGURE 9: PREDICTED AND MEASURED KINETIC ENERGY IN SECONDARY FLOW IN A CASCADE
LAKSHMINARAYANA (REF. 20)

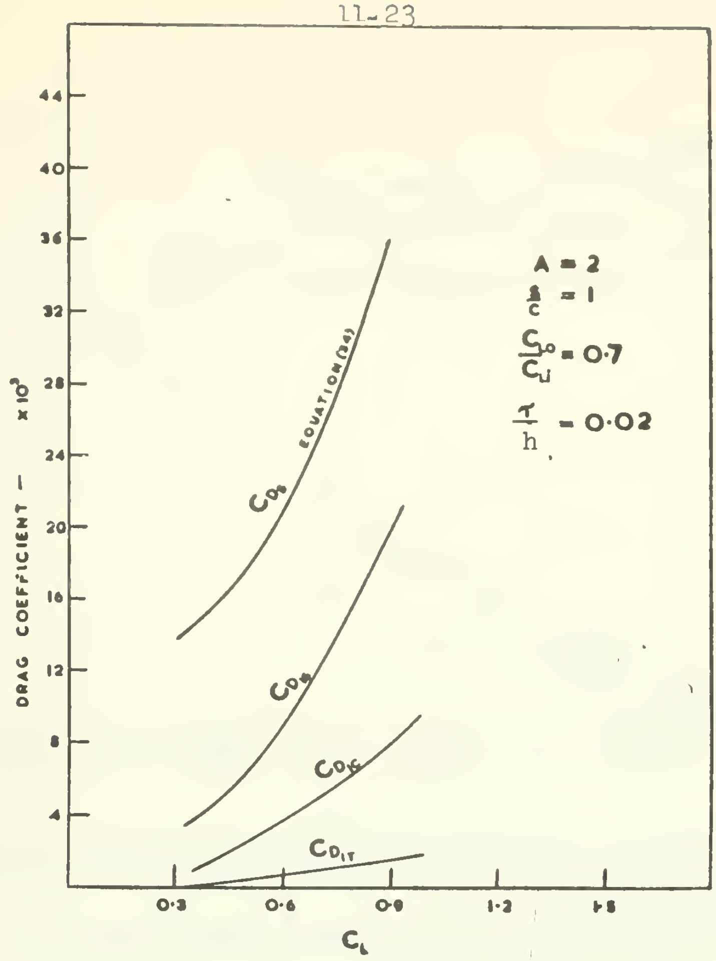


FIGURE 10 TOTAL SECONDARY LOSSES FOR A COMPRESSOR WITH A = 2, σ = 1
LAKSHMINARAYANA (REF. 21)

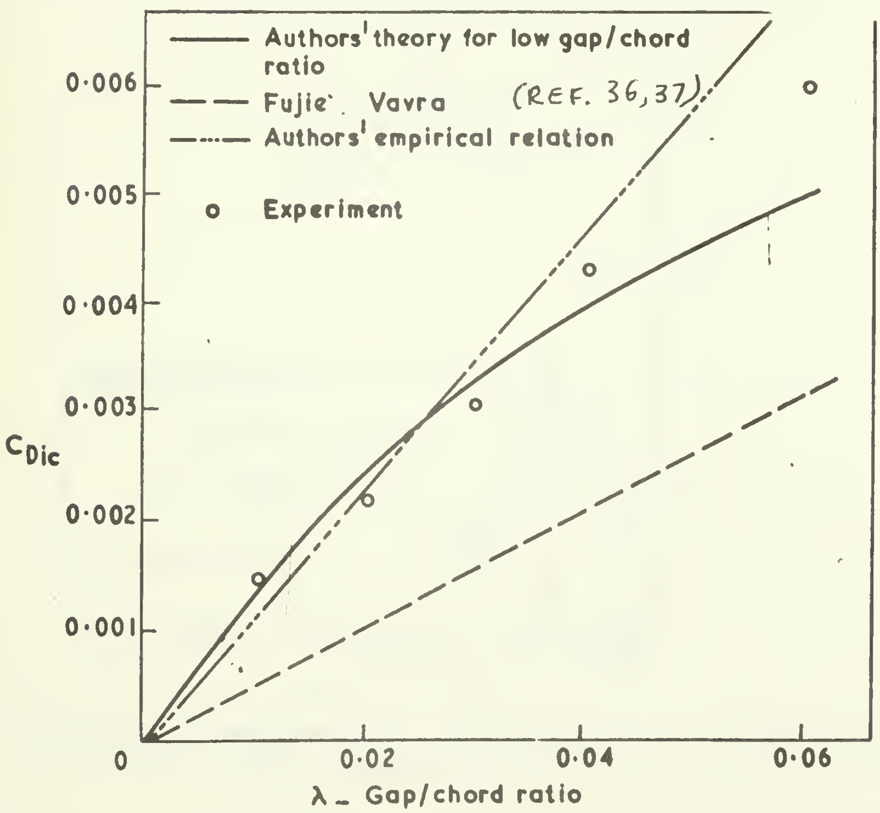


FIGURE 11: COMPARISON BETWEEN EXPERIMENTAL AND THEORETICAL VALUES OF C_{DIC} WITH UNIFORM INLET FLOW IN A CASCADE
LAKSHMINARAYANA (REF. 19)

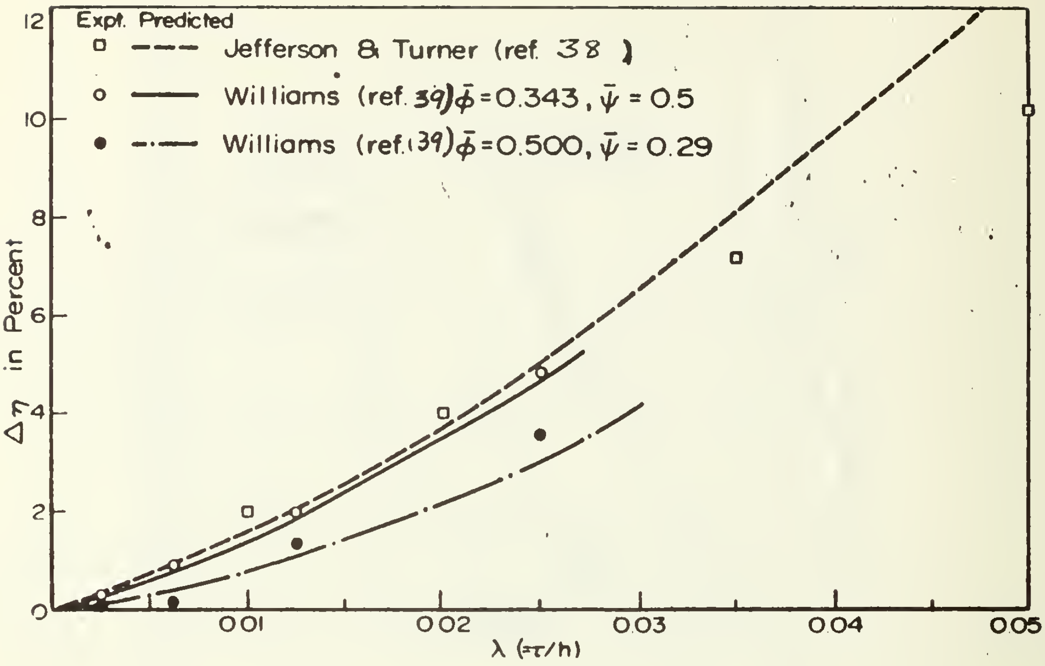


FIGURE 12: DECREASE IN STAGE EFFICIENCY WITH CLEARANCE FOR AXIAL FLOW COMPRESSORS
LAKSHMINARAYANA (REF. 23)

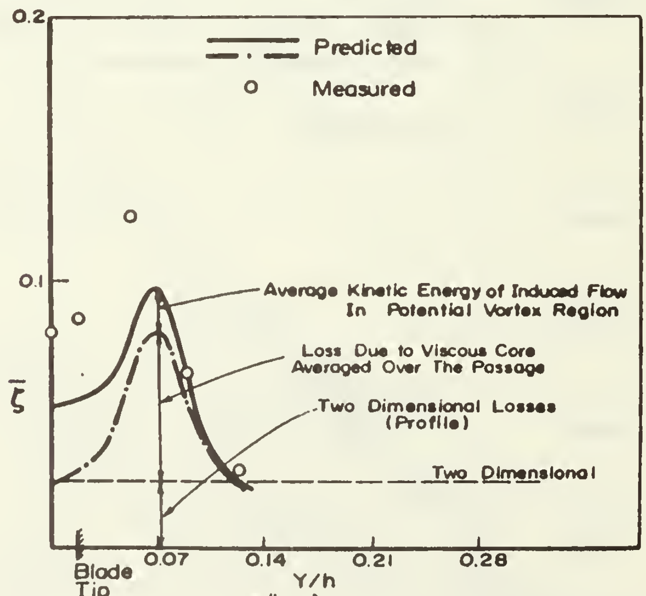


FIGURE 13: MEASURED AND PREDICTED LOSS COEFFICIENT FOR AUTHOR'S CASCADE
LAKSHMINARAYANA (REF. 23)

LOSS EVALUATION METHODS

Discussion Leader: Prof. B. Lakshminarayana

Presentation: The Turbulence Structural Hypothesis and Loss Coefficient Predictions

by

Dr. G. David Huffman

A NOTE ON LOW REYNOLDS NUMBER TURBULENT FLOWS**

by

G. DAVID HUFFMAN* AND PETER BRADSHAW
DEPARTMENT OF AERONAUTICS, IMPERIAL COLLEGE, LONDON

----

*Current address: Detroit Diesel Allison Division, General Motors Corp. Indianapolis, Indiana, U.S.A.

**Editor's Comment: This is the paper on which Dr. Huffman based his presentation.

A NOTE ON LOW REYNOLDS NUMBER TURBULENT FLOWS

by

G. David Huffman* and Peter Bradshaw

Department of Aeronautics, Imperial College, London

An analysis of existing data in low-Reynolds-number flows strongly suggests that the conclusion of Simpson (1970) concerning the variation of von Karmen's constant, κ, with Reynolds number is not correct. This implies that Coles' (1962) assumption of the validity of the logarithmic velocity profile at low Reynolds numbers is correct and, moreover, the inference drawn by Coles and later authors regarding the presence of viscous effects in the outer layer is valid. The analysis shows that these viscous effects are not present in duct flows which implies that they are associated with the "viscous superlayer"

It appears that the "viscous sublayer" is more strongly affected by shear-stres gradients or transverse wall curvature than is the rest of the inner layer.

---

1. Introduction

Coles (1962, 1968) analyzed the great bulk of available low Reynolds number and zero pressure gradient turbulent boundary layer measurements. He determine the surface shear stress from the velocity profile by utilizing the assumption that the velocity in the inner layer ($y/\delta \lesssim 0.2$) but outside the viscous sublaye ($u_\tau y/\nu \gtrsim 40$) followed the usual logarithmic form, i.e.,

$$\frac{u}{u_\tau} = \frac{1}{\kappa} \log \frac{u_\tau y}{\nu} + C \tag{1}$$

where $\kappa = 0.41$ and $C = 5.0$. Furthermore, Coles found that the velocity defect

$$\frac{u_e - u}{u_\tau} = f_1 \, (y/\delta) \tag{2}$$

- - - - - - - - - - - - - - - -

* Current address: Detroit Diesel Allison Division, General Motors Corporation, Indianapolis, Indiana, U.S.A.

was a function of the Reynolds number in the outer part of the boundary layer ($y/\delta \gtrsim 0.2$) for momentum thickness Reynolds numbers of less than about 5,000. In particular, the profile parameter, $(\kappa/2)\ \Delta u/u_\tau$ where Δu is defined in Figure 1, falls from about 0.6 at high Reynolds numbers to zero for Reynolds numbers of approximately 500. Note that Coles' 1968 analysis of the data of Wieghardt--which appears to be representative of the low Reynolds number data in general--gives slightly different results from the 1962 analysis and is presumably to be regarded as superseding the latter. Coles briefly discussed the validity of the logarithmic law and demonstrated that the surface shear stress deduced from it was within about 10% of that deduced from the momentum integral equation at least in the case of the more reliable experiments.

The modern derivation of the "mixing length" formula from which equation (1) follows by integration when $\tau/\rho = u_\tau^2 = \tau_w/\rho$ utilizes the assumption that the turbulent structure of the flow near the surface is unaffected by the flow further removed from the surface [Townsend (1961)]. Moreover, this formula seems to be valid over a wide range of outer layer conditions. The outer layer and outer boundary conditions are transmitted to the inner layer via the shear stress gradient $\partial\tau/\partial y$ which is non-zero when the flow and/or pressure gradient are functions of x. Townsend (1961) discusses certain anomalies in the turbulence structure (the "inactive motion") but concludes that these effects are primarily limited to high Reynolds numbers and should not alter the "mixing length". The validity in the inner layer of the mixing length formula and the resulting logarithmic law is generally accepted while the source of the supposed outer layer viscous effects is primarily speculative.

Simpson (1970) questioned the validity of the logarithmic law showing that his own data indicated that κ varied as $Re_\theta^{-1/8}$ when $Re_\theta < 6{,}000$. His velocity profiles collapsed together--except in the viscous sublayer--when plotted as u/u_e versus y/δ. [Compare this with equation (2) on which high Reynolds number data are expected to collapse]. The aforementioned variation of κ and likewise of C follows from the approximation $c_f \propto Re_\theta^{-1/4}$. Simpson measured the surface shear stress directly from the velocity gradient at the wall and indirectly from $d\theta/dx$ obtaining good agreement.

Recently Cebeci and Mosinskis (1970) have used values of κ and C varying with Re_θ as part of the input to a method of calculating turbulent boundary layers and showed improved agreement with experimental data. On the other hand, Herring and Mellor (1968) using a similar calculation method obtained improved agreement by altering the outer layer eddy viscosity assumptions--leaving κ and C unchanged. It seems that the presently available boundary layer data are not accurate enough to ascertain the validity of the logarithmic law.

In this paper, an analysis of data for flows in which Reynolds number effects on the inner layer are likely to be stronger than in a boundary layer and thus easier to detect is presented. The results show that C or its equivalent is Reynolds number dependent and that κ appears to be a constant to good accuracy. It appears that even the variation of C is likely to be small in boundary layers unless the influence of the outer layer is extremely large.

Accepting this as the best vindication of the logarithmic law that current data are likely to provide, it is shown that duct velocity defect profiles do follow equation (2) at low Reynolds numbers even though the boundary layer defect profiles do not. This paradox is at once resolved if it is supposed that the Reynolds number effects in the boundary layer are associated with the viscous superlayer [Corrsin and Kistler (1955)]. A conservative estimate of the superlayer thickness suggests that it may occupy a large fraction of the outer layer at low Reynolds numbers.

In the last sections of the paper, the status quo is restored by presenting an incidental result from the present analysis showing that transverse curvatur affects the viscous sublayer appreciably even when the ratio of sublayer thickness to radius of curvature is as small as 0.1. The flow in the inner layer but outside the sublayer is apparently unaffected by transverse curvature Therefore, the inner layer has been acquitted of violating the logarithmic law at low Reynolds numbers only to find evidence for its misbehavior in other circumstances--admittedly less important ones.

2. Analysis of Velocity Profiles in Low Reynolds Number Flows

The assumption that the turbulent flow near a smooth solid surface (y/δ or $y/a < 0.2$) depends only on u_τ, y, ρ, and ν leads via dimensional analysis to

$$u = u_\tau \, f_2 \, (\zeta)$$

where

$$\zeta = \frac{u_\tau y}{\nu} \quad (1 + \frac{y}{a})^i \tag{4}$$

and f_2 is a universal function of ζ, a the radius of the bounding surface (see Figure 2) and i takes on the values 0 and 1 for two-dimensional and axis-symmetric flow fields, respectively. The axis-symmetric formulation is presented by Willmarth and Yang (1970).

Equations (3) and (4) in conjunction with the momentum and continuity relations, i.e.,

$$\rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} = - \frac{dp}{dx} + \frac{1}{(a + y)^i} \frac{\partial}{\partial y} \left[(a + y)^i \, \tau\right] \tag{5}$$

and

$$\frac{\partial}{\partial x} \left[(a + y)^i \rho u\right] + \frac{\partial}{\partial y} \left[(a + y)^i \rho v\right] = 0 \tag{6}$$

yield the following shear stress equation

$$(1 + \frac{y^+}{a^+})^i \, \tau^+ = 1 + \pi^+ y^+ \, (1 + \frac{y^+}{2a^+})^i + v^+_w \, (u^+ - u^+_w) +$$

$$\frac{d(\ln u_\tau)}{dx^+} \int_0^{y^+} u^{+2} \, (1 + \frac{y^+}{a^+})^i \, dy^+ \tag{7}$$

where $y^+ = yu_\tau/\nu$, $a^+ = au_\tau/\nu$, $\tau^+ = \tau/\tau_w$, $\pi^+ = (\nu/\rho u_\tau^3)\, dp/dx$, $v^+ = v/u_\tau$,

$u^+ = u/u_\tau$, $dx^+ = u_\tau dx/\nu$, and the subscript w denotes the wall condition, i.e., $y^+ = 0$. The outer layer-inner layer communication parameter $\partial\tau^+/\partial y^+$ can be obtained by differentiating equation (7) and is

$$(1 + \frac{y^+}{a^+})^i \frac{\partial\tau^+}{\partial y^+} = \pi^+ (1 + \frac{y^+}{2a^+})^i + \frac{i\pi^+ y^+}{2a^+} (1 + \frac{y^+}{2a^+})^{i-1} + v^+_w \frac{\partial u^+}{\partial y^+} +$$

$$\frac{d\ln u_\tau}{dx^+} u^{+2} (1 + \frac{y^+}{a^+})^i - \frac{i\tau^+}{a^+} (1 + \frac{y^+}{a^+})^{i-1} \tag{8}$$

Note that $\partial\tau^+/\partial y^+$ is a function of π^+, a^+, v^+_w, and $d\ln u_\tau/dx^+$, for y^+ = Constant

The shear-stress can be related to the velocity gradient via

$$\tau^+ = \frac{\partial u^+}{\partial y^+} + (L^+ \frac{\partial u^+}{\partial y^+})^2 \tag{9}$$

which can be justified by local equilibrium arguments for the turbulent portion of the inner layer. Note that $L^+ = Lu_\tau/\nu$ where L is the "mixing length" and is proportional to y in the inner layer. Equation (9) has been utilized in non-equilibrium inner layer regions, e.g., the viscous sublayer where significant energy transport by turbulent diffusion normal to the surface occurs, by Patanka and Spalding (1967), Cebeci and Smith (1968), Herring and Mellor (1968) and McDonald (1969) by modifying Van Driest's (1956) original relation to a "local equilibrium" formulation

$$L^+ = \kappa y^+\{1-\exp[-(\tau^+)^{\frac{1}{2}} y^+/A^+]\}. \tag{10}$$

When $y^+ > 40$, $\tau^+ \approx 1$, $(L^+ \frac{\partial u^+}{\partial y^+})^2 >> \frac{\partial u^+}{\partial y^+}$, and $L^+ \approx \kappa y^+$, equations (9) and (10) yield equation (1) where $A^+ = 26$ corresponds approximately to $C = 5$.

In the present analysis, the integral of

$$\frac{\partial u^+}{\partial y^+} = \frac{(1 + 4L^{+2}\tau^+)^{\frac{1}{2}} - 1}{2L^{+2}} \tag{11}$$

along with equations (7) and (10) have been fitted to a number of experimental velocity profiles. κ and A^+ have been chosen to give the best fit in the region between the surface and the expected reliability limit of equations (7), (10), and (11). It was established fairly soon that the optimum value of κ was about 0.41. The optimum value of A^+ was found numerically by means of a logical search algorithm described by Huffman (1971) and Huffman, Jones and Brodkey (1971).

Figures 3 through 10 show the results of the data analysis plotted in terms of u^+ and y^+ where

$$u^+ = f_3(y^+, \langle\partial\tau^+/\partial y^+\rangle) \tag{12}$$

and

$$\left\langle \frac{\partial\tau^+}{\partial y^+} \right\rangle = \frac{\int_o^{y^+} (1 + \frac{y^+}{a^+})^i \left(\frac{\partial\tau^+}{\partial y^+}\right) dy^+}{\int_o^{y^+} (1 + \frac{y^+}{a^+})^i \, dy^+} \tag{13}$$

The effect of external conditions on u^+ is transmitted by either $\langle\partial\tau^+/\partial y^+\rangle$ or alterations in the u^+ boundary condition. Again $\langle\partial\tau^+/\partial y^+\rangle$ depends on π^+, a^+, v_w^+, and/or $d\ln u_\tau/dx^+$.

The data of Patel and Head (1969) dealing with low Reynolds number flows in circular pipes and two-dimensional ducts is shown in Figures 3, 4, and 5. In this case, the dimensionless shear stress gradient is constant and equals $-1/a^+$ where a^+ is the radius of the pipe or duct half-width. The duct Reynolds number based on the mean velocity, i.e., $2a\bar{u}/\nu$, can be related to $\langle\partial\tau^+/\partial y^+\rangle$ and is approximately $-30/\langle\partial\tau^+/\partial y^+\rangle$. It can be seen that equation (12) is truly independent of inner layer arguments since it simply states that the dimensionless velocity is a function of Reynolds number and position. Moreover, the inner layer arguments are sufficiently well established for us to accept $\partial\tau^+/\partial y^+$ as representing the major effect of external conditions on the inner layer even when $\partial\tau^+/\partial y^+$ is not uniquely determined by the Reynolds number. This behavior is indeed exhibited and in Figures 6, 7, 8, 9, and 10, $\partial\tau^+/\partial y^+$ is at times completely independent of the Reynolds number.

It is to be expected that A^+ will be a universal function of $\partial\tau^+/\partial y^+$. If it can be shown that A^+ is virtually constant for small values of $\partial\tau^+/\partial y^+$ which characterize constant pressure boundary layers, then the logarithmic law as used by Coles is re-established. Figure 14 shows the variation of A^+ for a series of two-dimensional flows, i.e., wall jets, channel flows, and boundary layers. Since $-\partial\tau^+/\partial y^+$ is about 10^{-3} in the inner layer of a boundary layer for $Re_\theta \approx 1{,}000$ and numerically less at higher Reynolds numbers, it can be concluded that A^+ will be virtually constant in boundary layers where $Re_\theta >$ 1,000 and thus the logarithmic relationship holds.

3. Behavior of the Outer Layer

It can be seen from Figures 11, 12, and 13 that the velocity defect profiles in a pipe are virtually unaffected by Reynolds number outside the viscous sublayer [noting that equation (10) implies a constant value of $(\tau^+)^{\frac{1}{2}}\ y^+$ rather than y^+ for the edge of the sublayer if the two are different]. At the lower Reynolds numbers, the viscous sublayer is so thick that the region of collapse is very small; however, noting that Re_θ is only about 0.07 of $2\bar{u}a/\nu$, or $-2/\langle\partial\tau^+/\partial y^+\rangle$, it is apparent that the outer layer in a pipe or duct is unaffected by viscosity at Reynolds numbers much less than those causing marked changes in the boundary layer "wake component".

Accepting the validity of the logarithmic law, implies that outer layer Reynolds numbers effects first noted by Coles (1962) are indeed real. The presence of these effects in boundary layers and their absence from pipe and duct flows must be connected with the presence of an interface between the turbulent and irrotational flow in the former and its absence in the latter case. This interface contains the viscous superlayer wherein mean and fluctuating vorticity are communicated by viscous action to previously irrotational fluid.

The average thickness of the superlayer , δ_{sup}, is expected to depend--at least to a first approximation--on the mean-square vorticity in the turbulence near the superlayer, $\overline{\omega^2}$, and on the viscosity itself. In locally isotropic turbulence, $\overline{\omega^2} = \varepsilon/\nu$ where ε is the energy dissipation rate so that δ_{sup} must be approximately proportional to the Kolmogorov length scale $\eta = (\nu^3/\varepsilon)^{\frac{1}{4}}$ near the inner boundary of the superlayer. The ratio of the viscous sublayer thickness, δ_{sub}, to the value of η at $y^+ \approx 40$ is approximately 20; consequently, δ_{sup} may be significantly larger than the local η which in turn will be significantly larger than η near the sublayer because ε is less near the superlayer than near the sublayer. A conservative conclusion is that the superlayer is not much thinner than the sublayer.

The sublayer is plane whereas the superlayer is distributed over a highly irregular interface. Indeed, the irregularity of the interface seems to increase at low Reynolds numbers judging by the intermittency measurements and smoke photographs of Fiedler and Head (1966). Consequently, the fraction of the boundary layer fluid that is occupied by the superlayer is much larger than δ_{sup}/δ. If δ_{sup}/δ is of the same order as δ_{sub}/δ, it is approximately 0.015 at $Re_\theta = 5{,}000$ and 0.1 at $Re_\theta = 600$. Clearly, the superlayer may have a large influence on the outer layer at low Reynolds numbers.

4. The Effect of Transverse Curvature

The differences in the values of A^+ for two-dimensional and axis-symmetric flow fields (see Figure 15) combined with the apparent equality of κ, suggest that transverse curvature may affect the viscous sublayer but not the remaining portion of the inner layer. The value of $<\partial\tau^+/\partial y^+>$ at which the A^+ values start to diverge corresponds to $\delta_{sub}/a \simeq 0.1$ whereas the ratio of inner layer thickness to a is about 0.2. There seemed to be no apparent reason for the observed viscous sublayer sensitivity to transverse curvature so some additional data were analyzed. These cases consisted of flows on concave as well as convex surfaces, i.e., flows on the outside of circular cylinders. These velocity fields are plotted in Figures 6, 7, and 8. Additional two-dimensional boundary layer data is shown in Figures 9 and 10.

The agreement between the measured and calculated velocities in Figures 6, 7, 8, 9, and 10 is not as good as that of Figures 3, 4, and 5; however, it is of sufficient accuracy to ascertain the general trend of the A^+ - $\partial\tau^+/\partial y^+$ relation. The annular duct, axis-symmetric wall jet, and axis-symmetric boundary layer data clearly show a trend of A^+ with $\partial\tau^+/\partial y^+$ in the opposite sense to that found with pipes. Note that $\partial\tau^+/\partial y^+$ is dependent on the flow field transverse curvature, e.g., see equation (8), and is thus a suitable curvature parameter itself.

There are no obvious explanations of the apparent curvature effect other than it is a real effect of curvature on the sublayer. The consistency of the trend from pipe to duct to annulus seems to rule out arguments based on differences between any two of these flows. A possible clue to the surprisin sensitivity of the sublayer to transverse curvature comes from the observatio by Kline et. al. (1967) of a tendency to transverse periodicity in the sublay a wavelength λ given by $u_\tau\lambda/\nu = \tau^+ \approx 100$. Moreover, $\lambda/a \approx 3\ \delta_{sub}/a \approx 100$ $(\partial\tau^+/\partial y^+)$; consequently, λ/a is about 0.3, i.e., one transverse wavelength subtends about 20°, when significant curvature effects begin. This transvers scale is quite large when compared to the eddy length scales just outside the viscous sublayer, i.e., $L^+_{sub} \approx 16$ when $y^+ \approx 40$, so that it is not implausible tht the sublayer is affected while the remainder of the inner layer is not.

5. Conclusions

The main object of the data analysis of Section 2 was to demonstrate the validity of the logarithmic velocity profile law at values of $\partial\tau^+/\partial y^+$ found in constant pressure boundary layers. $\partial\tau^+/\partial y^+$ is the only externally-imposed parameter on which the inner layer velocity profile can depend according to the accepted local equilibrium arguments [Townsend (1961)]. While gross external disturbances might violate these conditions, one would still expect the first effect of external disturbances to appear via $\partial\tau^+/\partial y^+$. It is, therefore, safe to conclude that the logarithmic law is valid in a boundary layer at all Reynolds numbers greater than those for which reverse transition occurs ($Re_\theta \approx 350$).

If the validity of the logarithmic law is accepted, it appears that the data analysis of Coles (1962) implies viscous effects in the outer part of a boundary layer which are not present in the core of a pipe or channel flow and, thus, are associated with the viscous superlayer.

Departures from the logarithmic law do occur at large negative values of $\partial\tau^+/\partial y^+$ or $\langle\partial\tau^+/\partial y^+\rangle$. These changes in profile slope outside the viscous sublayer are well represented by utilizing the local rather than the wall value of the shear stress in the mixing length formula while keeping $\kappa \approx 0.41$. The profile within the viscous sublayer as well as the sublayer growth is well represented by making the "damping constant" A^+ a function of an "average" shear stress gradient, $\langle\partial\tau^+/\partial y^+\rangle$.

The $A^+ - \langle\partial\tau^+/\partial y^+\rangle$ relationship is different for pipes, plane flows, and annular flows and the limited data suggest that the difference is directly attributable to transverse surface curvature and not to some less obvious effect. The transverse curvature effect is limited to the sublayer. This may be due to long-wavelength transverse periodicity in the sublayer resulting from surface curvature [see Kline et. al. (1967)]. In the sublayer, the transverse length scale greatly exceeds the mixing length while the converse is true in the remainder of the inner layer. This would imply pronounced curvature effects in the sublayer and negligible effects elsewhere.

6. References

BARDI NARAYANAN, M. A. and RAMJEE, V. 1969 J. Fluid Mech., 35, 225

BRADSHAW, P. and GEE, M. 1962 ARC R & M 3252

CEBECI, T. 1968 Douglas Aircraft Corporation Paper 5524

CEBECI, T. and MOSINSKIS, G. J. 1970 "ASME Space Technology and Heat Transfer Conference Proceedings", Part 2.

CEBECI, T. and SMITH, A. M. O. 1968 Douglas Aircraft Division Report DAC 67130

COLES, D. E. 1962 The Rand Corporation Report R-403-PR

COLES, D. E. and HIRST, E. A. "Proceedings AFOSR-IFP-Stanford 1968 Conference on Turbulent Boundary Layer Prediction", Volume 2, Stanford

CORRSIN, S. and KISTLER, A. L. 1955 NACA Report 1244

HERRING, H. J. and MELLOR, G. L. 1968 NASA Contractor Report, NASA CR-1144.

HUFFMAN, G. D. 1971 Imperial College of Science and Technology IC Aero Report 71-06.

HUFFMAN, G. D., JONES, C. D. and BRODKEY, R. S. 1971 "Optimization of Turbulence Models by Means of a Logical Search Algorithm", submitted for publication

JULIEN, H. L., KAYS, W. M. and MOFFAT, R, J. 1970 Stanford University Department of Mechanical Engineering Report HMT-10.

KLINE, S. J., REYNOLDS, W. C., SCHRAUB, F. A. and RUNSTADLER, P. W. 1967 J. Fluid Mech., 30, 741

LAUFER, J. 1950 NACA TN 2123

LAUFER, J. 1954 NACA Report 1174

LAWN, C. H. 1968 CEGB RN RD/B/N 1232

MCDONALD, H. 1969 J. Fluid Mech., 35, 311

PATANKAR, S. V. and SPALDING, D. B. 1968 Heat and Mass Transfer in Boundary Layers, C.R.C. Press, Cleveland, 21

PATEL, V. C. and HEAD, M. R. 1969 J. Fluid Mech., 38, 181

SIMPSON, R. L. 1970 J. Fluid Mech., 42, 769

STARR, J. B. and SPARROW, E. M. 1967 J. Fluid Mech., 29, 495

TOWNSEND, A. A. 1961 J. Fluid Mech., 11, 97

VAN DRIEST, E. R. 1956 J. Aero. Sci, 23, 1007

WILLMARTH, W. W. and YANG, C. W. 1970 J. Fluid Mech., 41, 47

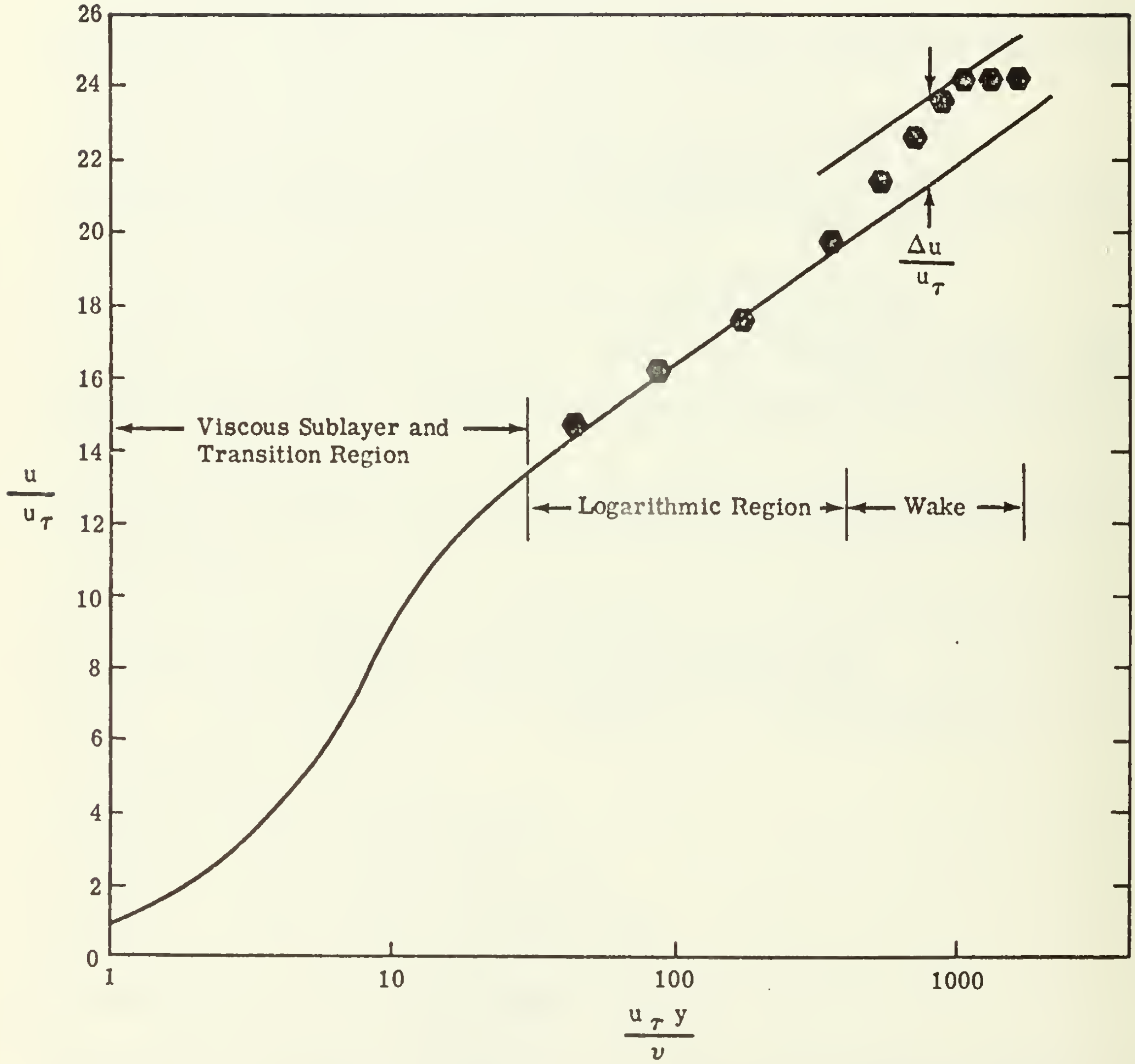


Figure 1. The "Law of the Wall" and "Law of the Wake" for a Typical Turbulent Boundary Layer.

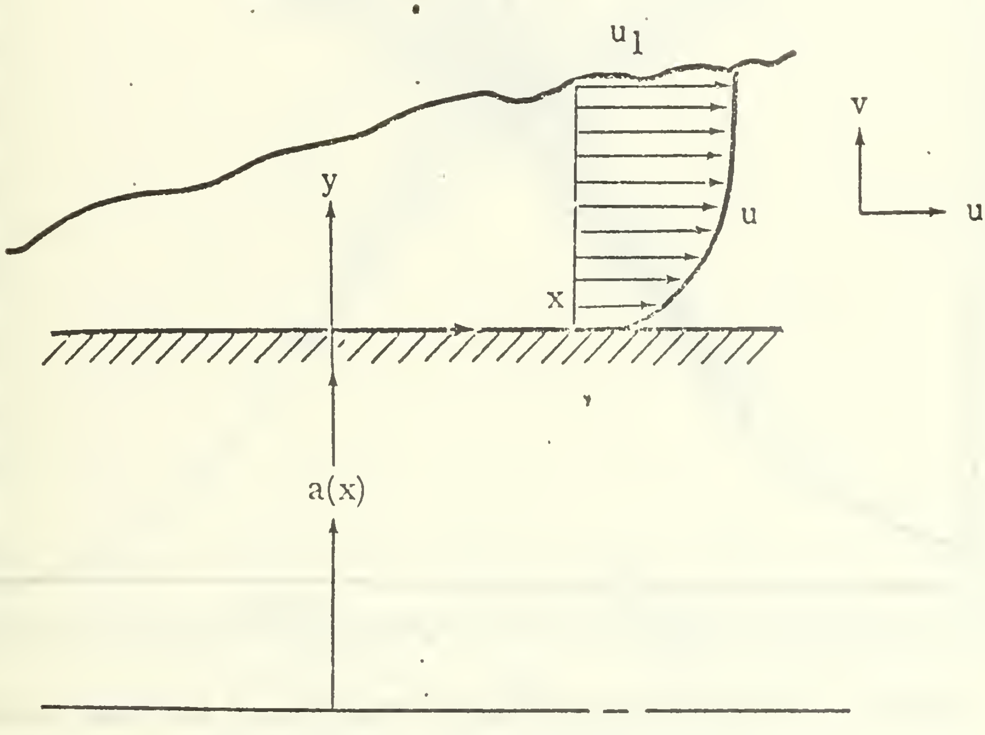


Figure 2. Flow Field Geometry.

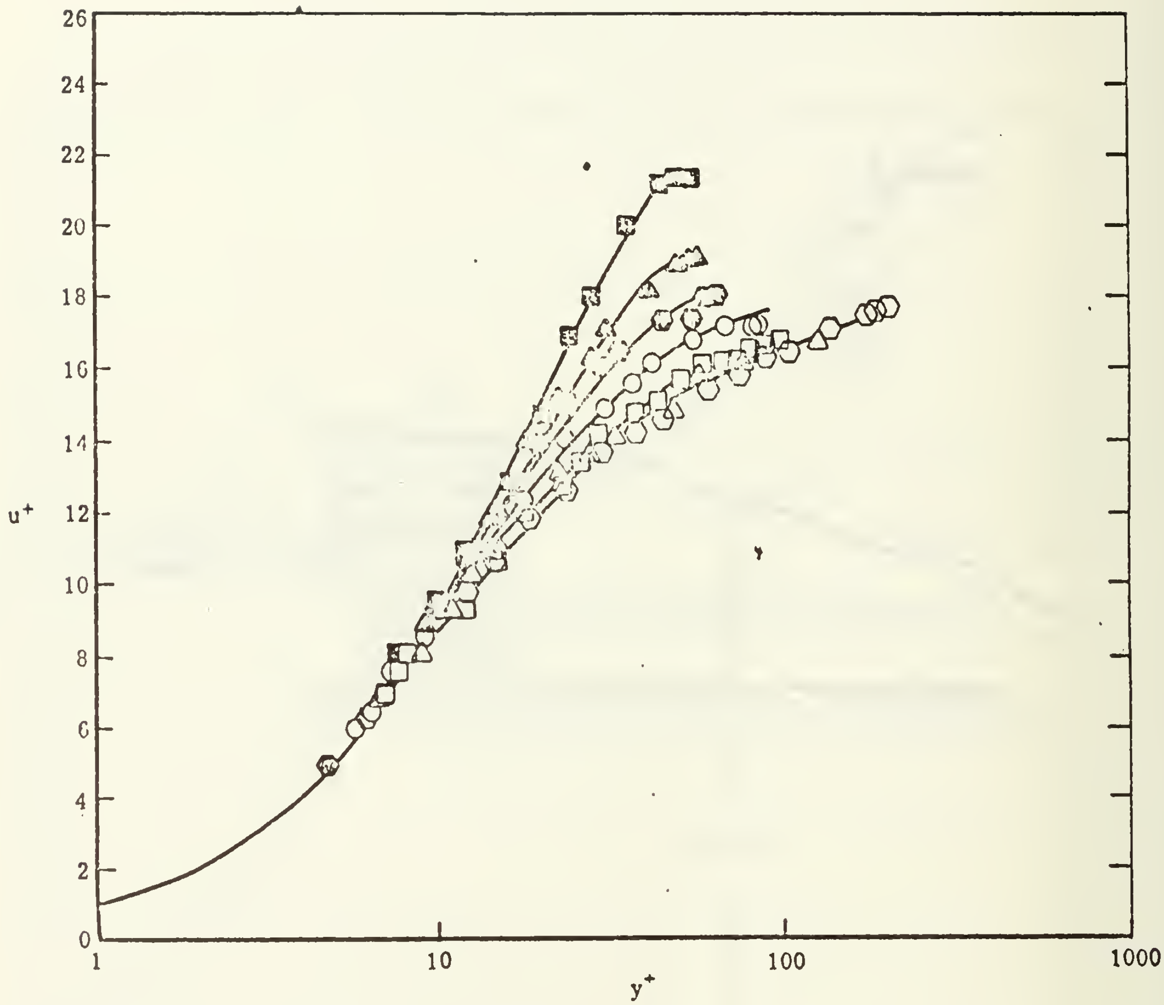


Figure 3. Fully-Developed Channel Flow [Patel and Head (1969)].
⬡ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0049$, $A^+ = 30.$; △ $\langle \partial\tau^+/\partial y^+ \rangle = -0.008$, $A^+ = 32.$; □ $\langle \partial\tau^+/\partial y^+ \rangle = -0.010$, $A^+ = 36.$, ○ $\langle \partial\tau^+/\partial y^+ \rangle = -0.013$, $A^+ = 45.$; ⬢ $\langle \partial\tau^+/\partial y^+ \rangle = -0.015$, $A^+ = 61.$; ▲ $\langle \partial\tau^+/\partial y^+ \rangle = -0.017$, $A^+ = 80.$; ■ $\langle \partial\tau^+/\partial y^+ \rangle = -0.019$, $A^+ = 140$; —— Computed velocities.

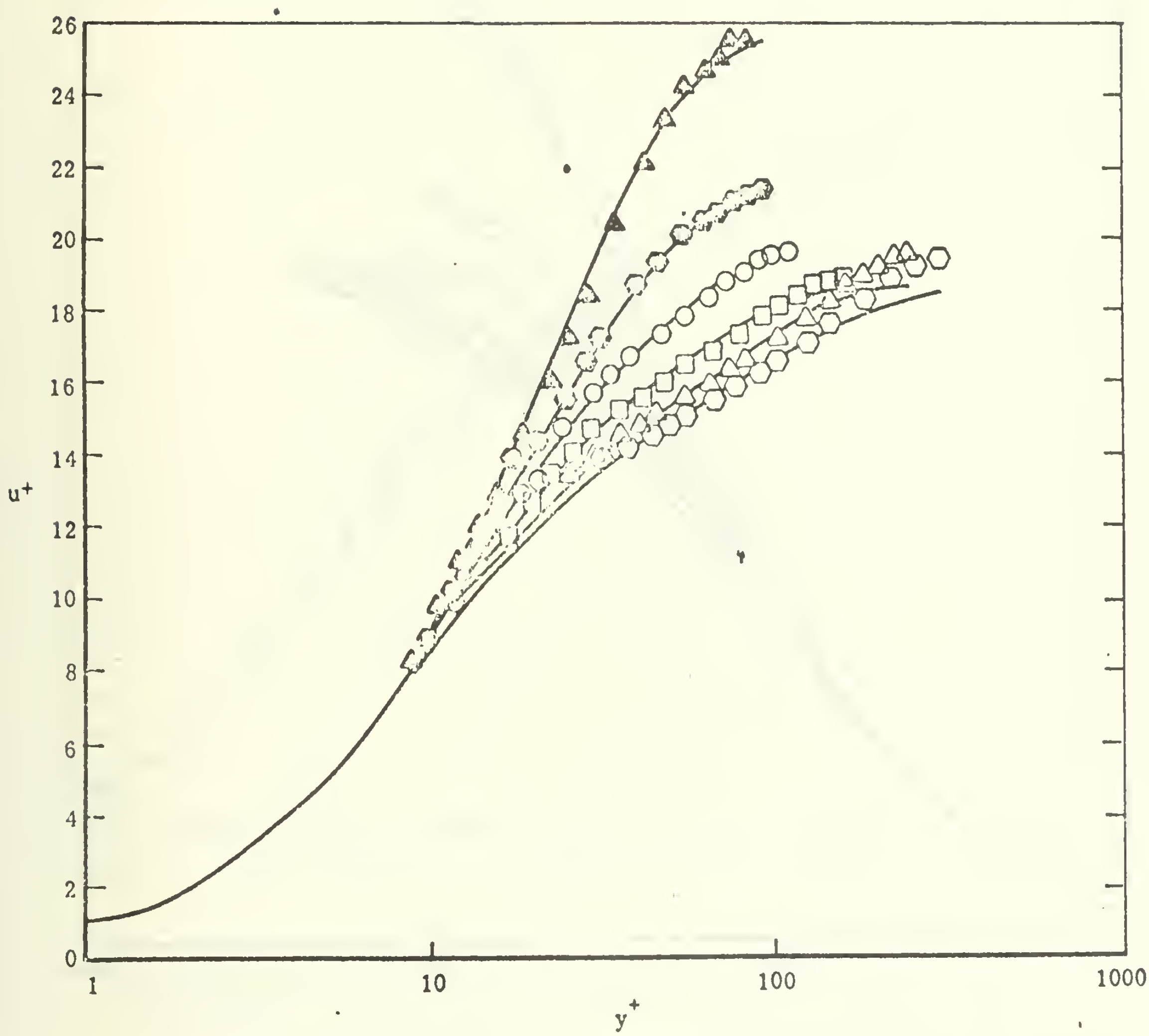


Figure 4. Fully-Developed Pipe Flow. [Patel and Head (1969)].
⬡ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0034$, $A^+ = 31$; △ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0042$, $A^+ = 34.$; □ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0065$, $A^+ = 40.$; ○ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0092$, $A^+ = 52$; ⬢ $\langle \partial\tau^+/\partial y^+ \rangle = -0.011$, $A^+ = 73.$; ▲ $\langle \partial\tau^+/\partial y^+ \rangle = -0.012$, $A^+ = 130$;—— Computed velocities.

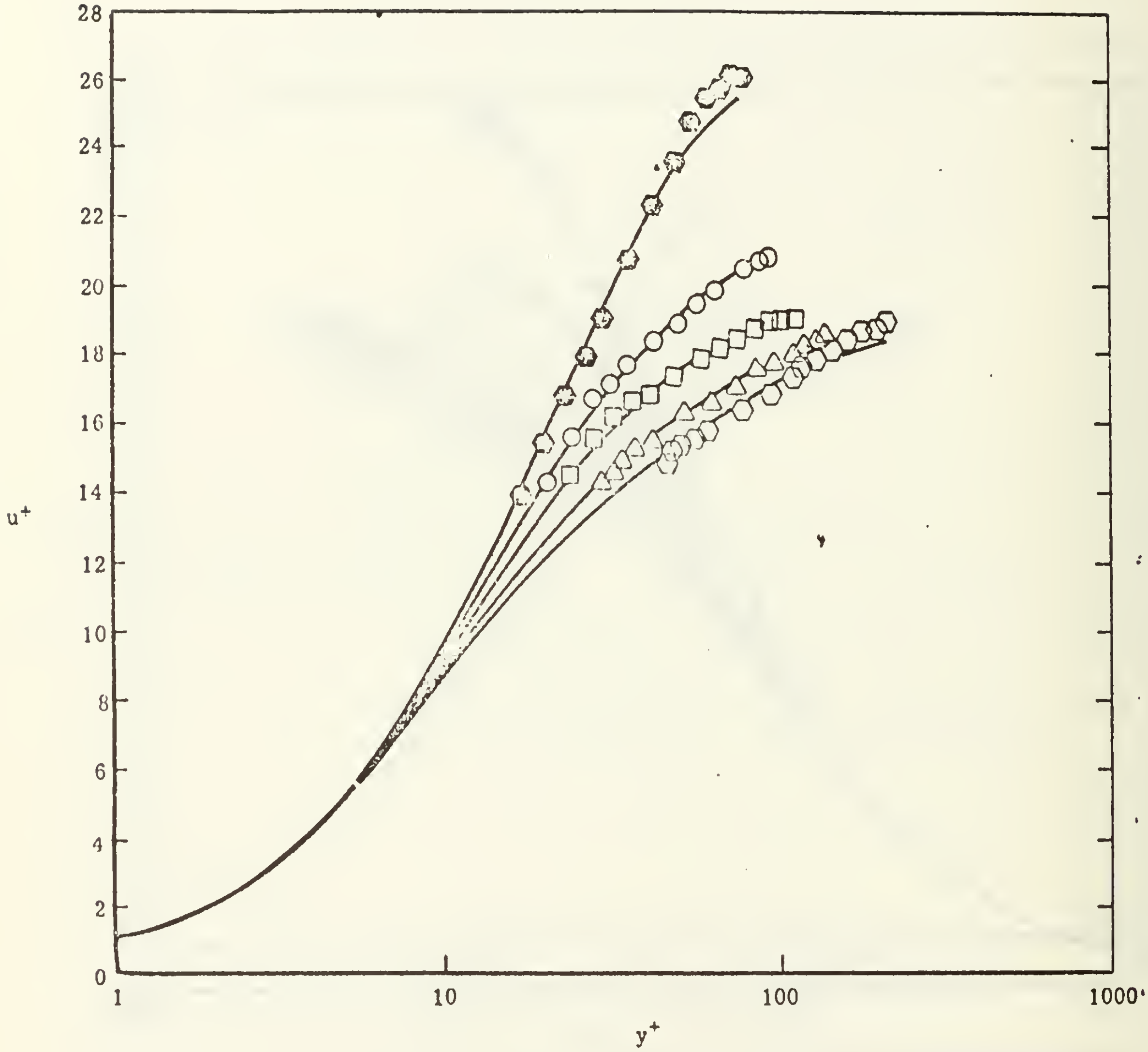


Figure 5. Fully-Developed Pipe Flow. [Patel and Head (1969)].
○ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0046_5$, $A^+ = 33.$; △ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0070$, $A^+ = 39.$; □ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0089$, $A^+ = 50.$; ○ $\langle \partial\tau^+/\partial y^+ \rangle = -0.010$; $A^+ = 67.$; ● $\langle \partial\tau^+/\partial y^+ \rangle = -0.012$, $A^+ = 130$;
—— Computed velocities.

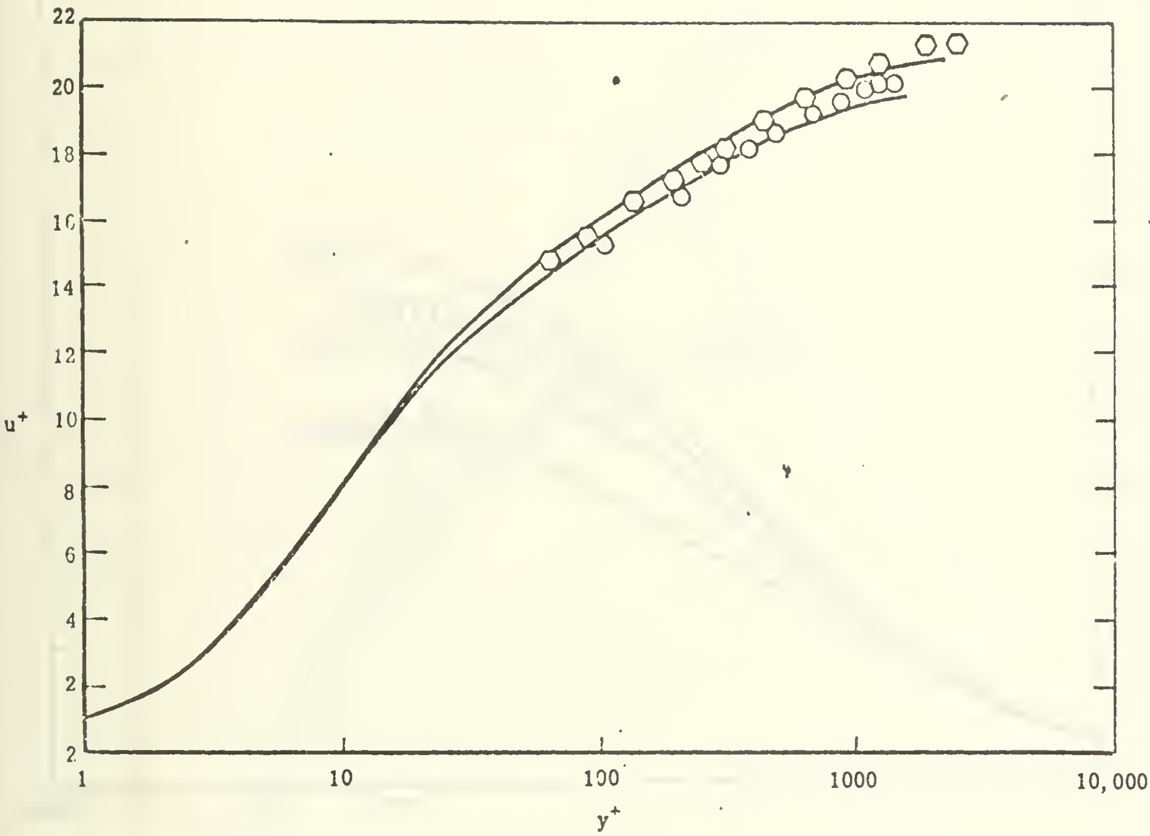


Figure 6. Fully-Developed Annular Flow. [Lawn (1968)].

⬡ $\langle \partial\tau^+/\partial y^+ \rangle = -\ 0.0015,\ A^+ = 25.,\ a^+ = 780;$

○ $\langle \partial\tau^+/\partial y^+ \rangle = -\ 0.0023,\ A^+ = 24.,\ a^+ = 480;$

——Computed velocities.

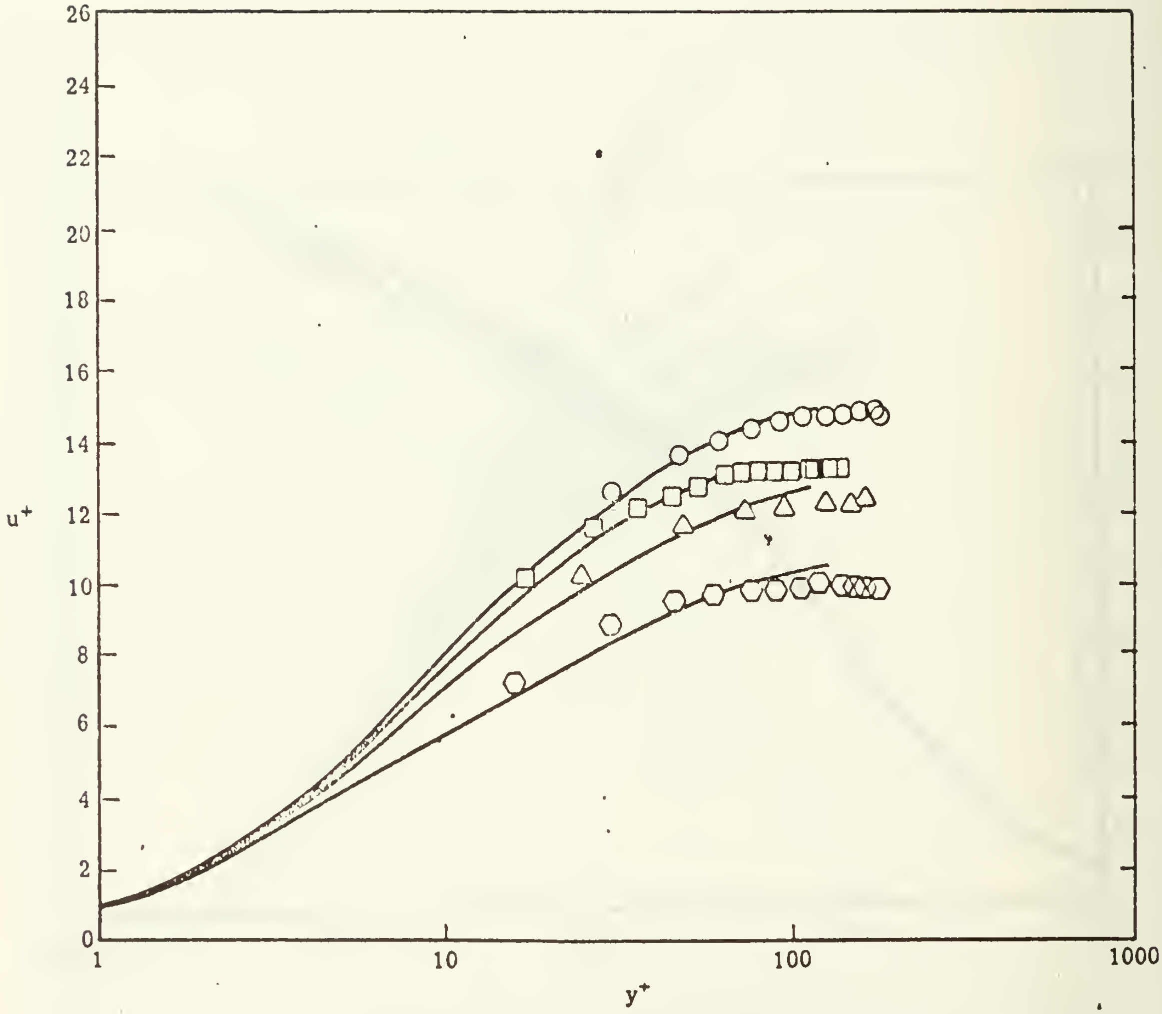


Figure 7. Axis-Symmetric Wall Jet. [Starr and Sparrow (1967)].

⬡ $<\partial\tau^+/\partial y^+> = -\ 0.0050,\ A^+ = 5.70,\ a^+ = 180;$

△ $<\partial\tau^+/\partial y^+> = -\ 0.0027,\ A^+ = 13.0,\ a^+ = 280;$

□ $<\partial\tau^+/\partial y^+> = -\ 0.0024,\ A^+ = 18.0,\ a^+ = 590;$

○ $<\partial\tau^+/\partial y^+> \doteq -\ 0.0018,\ A^+ = 21.0,\ a^+ = 1000;$

—— Computed velocities.

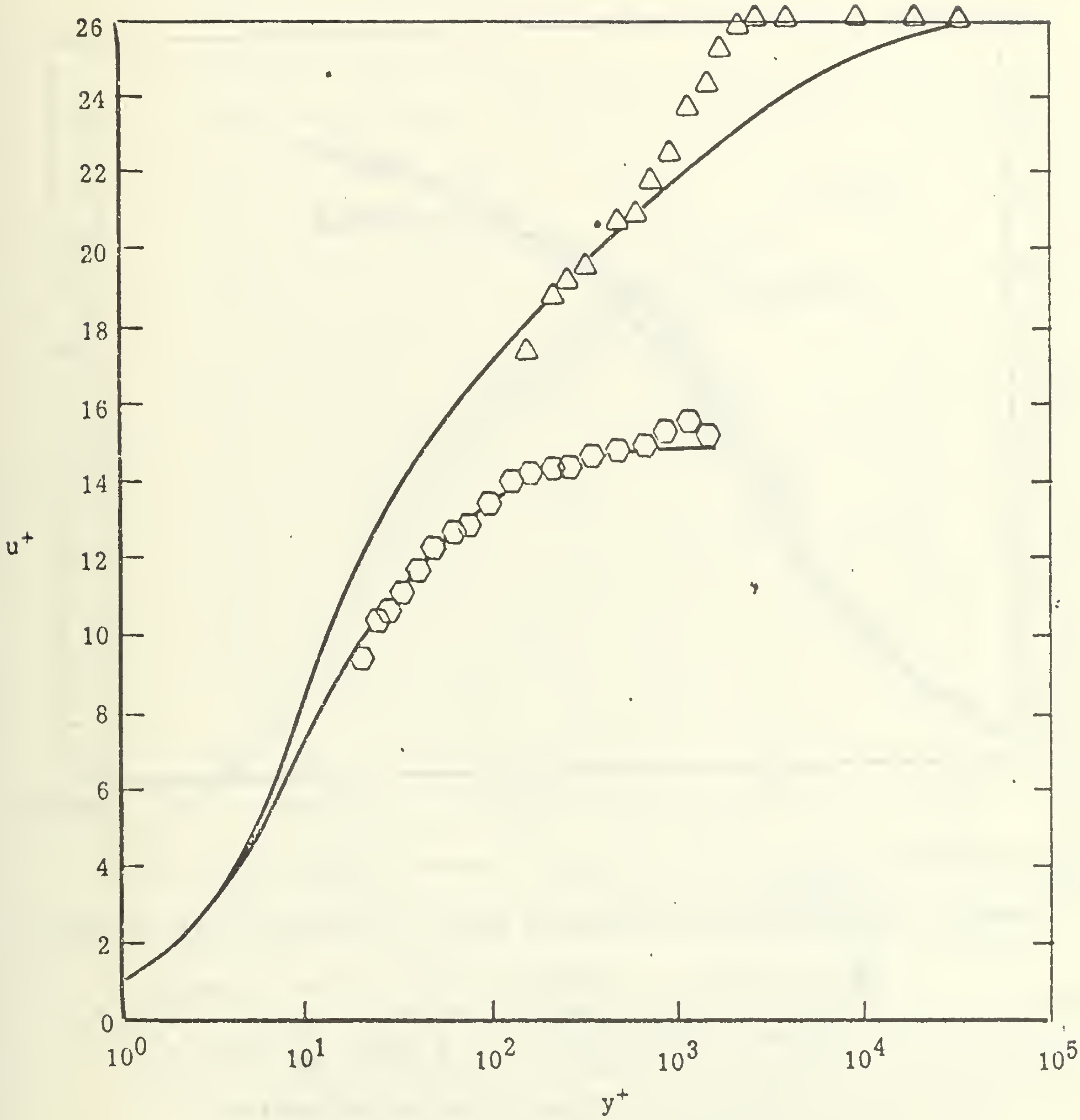


Figure 8. Axis-Symmetric Boundary Layer. [Cebeci (1968)].

△ $<\partial\tau^+/\partial y^+> = -\ 0.00062$, $A^+ = 27.$, $a^+ = 1500$;

○ $<\partial\tau^+/\partial y^+> = -\ 0.016$, $A^+ = 29.$, $a^+ = 17.$; —— Computed velocities.

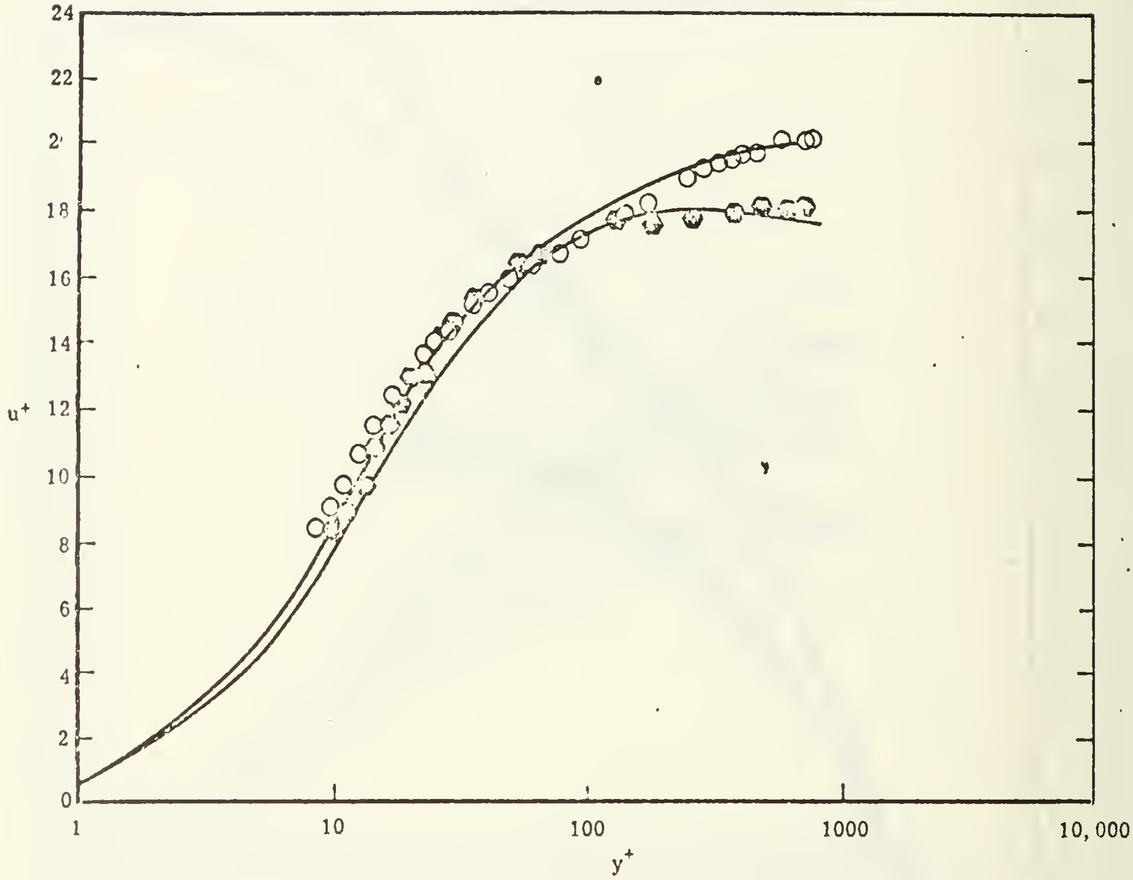


Figure 9. Two-Dimensional Boundary Layer. [Julian et. al. (1970)]. ● $<\partial\tau^+/\partial y^+> = -\ 0.018$, $A^+ = 67.$, $\pi^+ = -\ 0.0087$, $v^+_w = -\ 0.0047$; ○ $<\partial\tau^+/\partial y^+> = -\ 0.0096$, $A^+ = 40.$, $\pi^+ = -\ 0.012$, $v^+_w = 0.0$; ——Computed velocities.

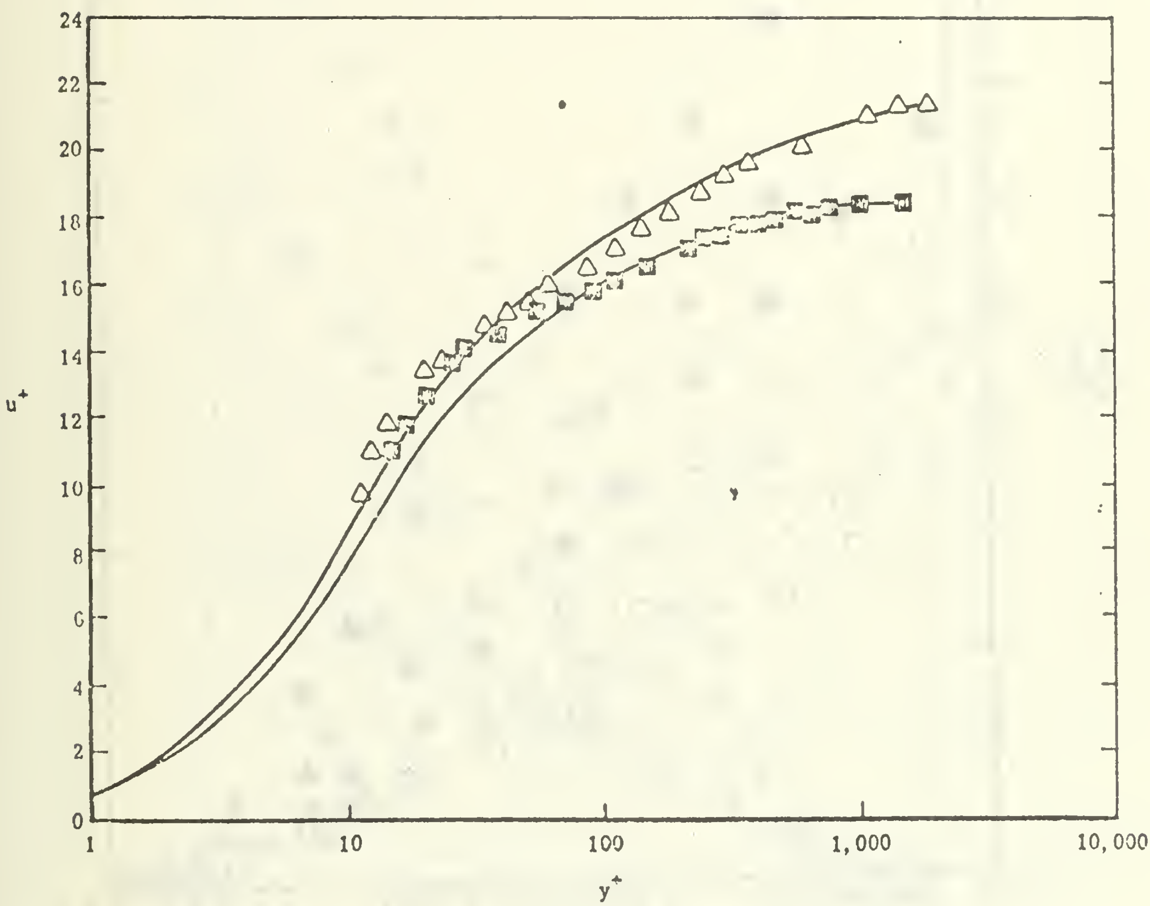


Figure 10. Two-Dimensional Boundary Layer. [Julian et. al. (1970)].

■ $<\partial\tau^+/\partial y^+> = -0.016$, $A^+ = 44.$, $\pi^+ = -0.0034$; $v^+_w = -0.037$; △ $<\partial\tau^+/\partial y^+> = -0.0048$, $A^+ = 35$, $\pi^+ = -0.0057$, $v^+_w = 0.0$; —— Computed velocities.

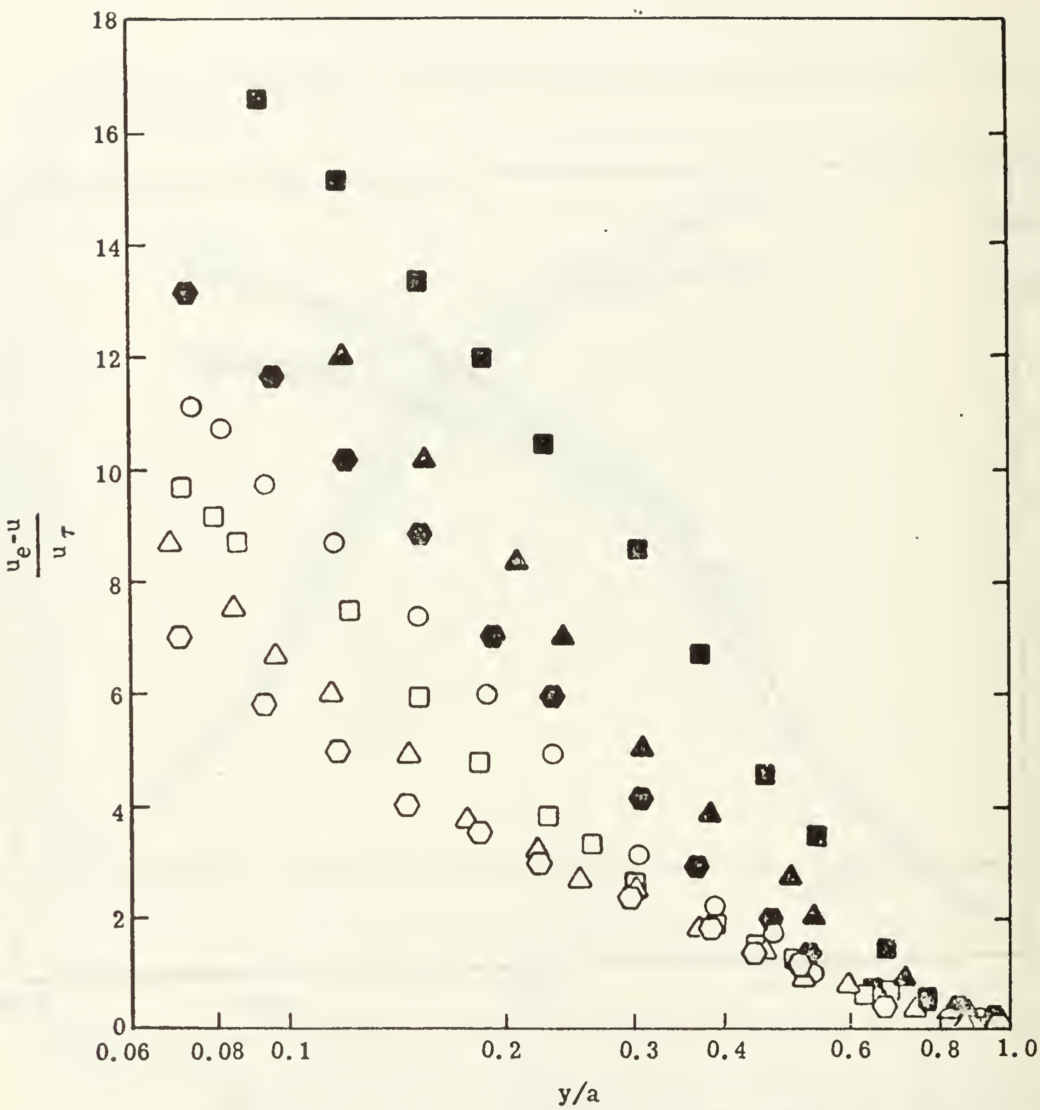


Figure 11. Fully-Developed Channel Flow [Patel and Head (1969)].
⬡ $<\partial\tau^+/\partial y^+> = -0.0049$; △ $<\partial\tau^+/\partial y^+> = -0.008$; □ $<\partial\tau^+/\partial y^+> = -0.010$; ○ $<\partial\tau^+/\partial y^+> = -0.013$; ⬢ $<\partial\tau^+/\partial y^+> = -0$
▲ $<\partial\tau^+/\partial y^+> = -0.017$; ■ $<\partial\tau^+/\partial y^+> = -0.019$.

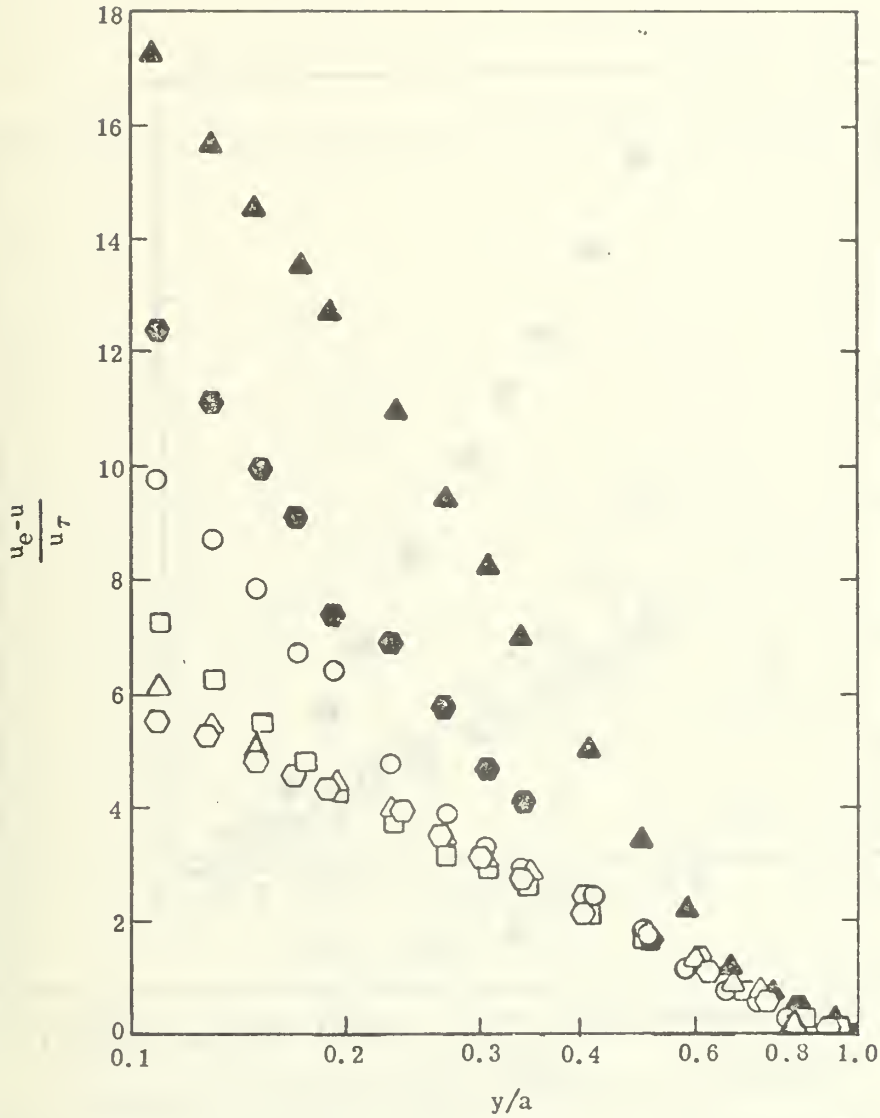


Figure 12. Fully-Developed Pipe Flow. [Patel and Head (1969)].

⬡ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0034$; △ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0042$;

□ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0065$; ○ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0092$;

⬢ $\langle \partial\tau^+/\partial y^+ \rangle = -0.011$; ▲ $\langle \partial\tau^+/\partial y^+ \rangle = -0.012$.

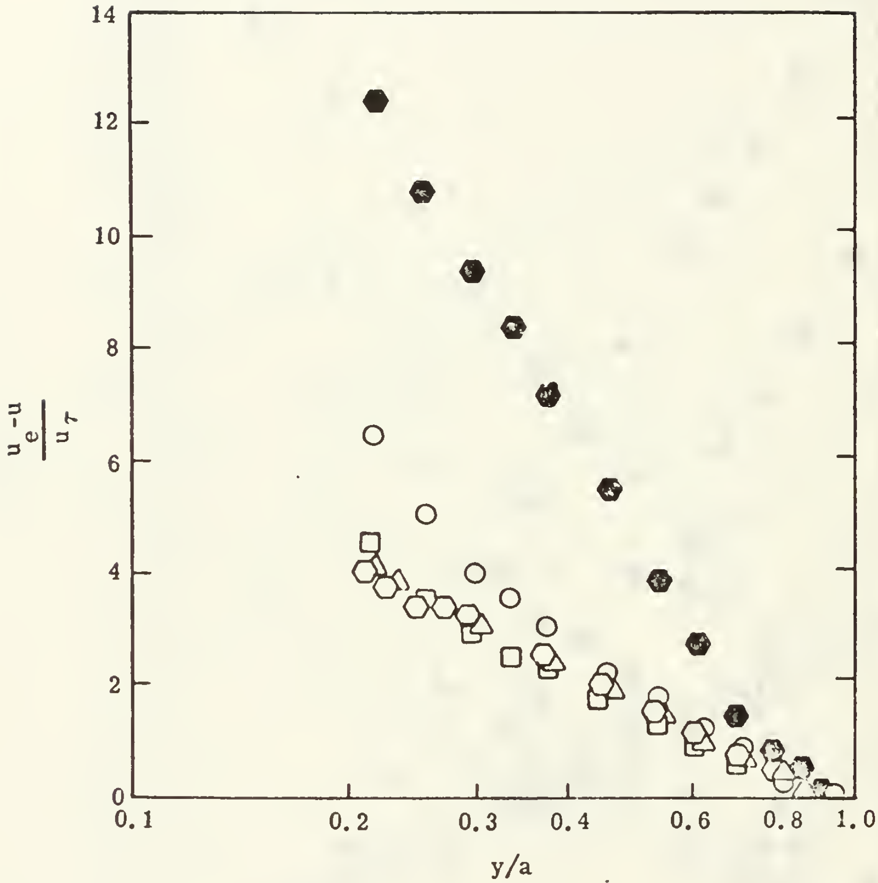


Figure 13. Fully-Developed Pipe Flow. [Patel and Head (1969)].

○ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0046$; △ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0070$;

□ $\langle \partial\tau^+/\partial y^+ \rangle = -0.0089$; ○ $\langle \partial\tau^+/\partial y^+ \rangle = -0.010$;

● $\langle \partial\tau^+/\partial y^+ \rangle = -0.012$.

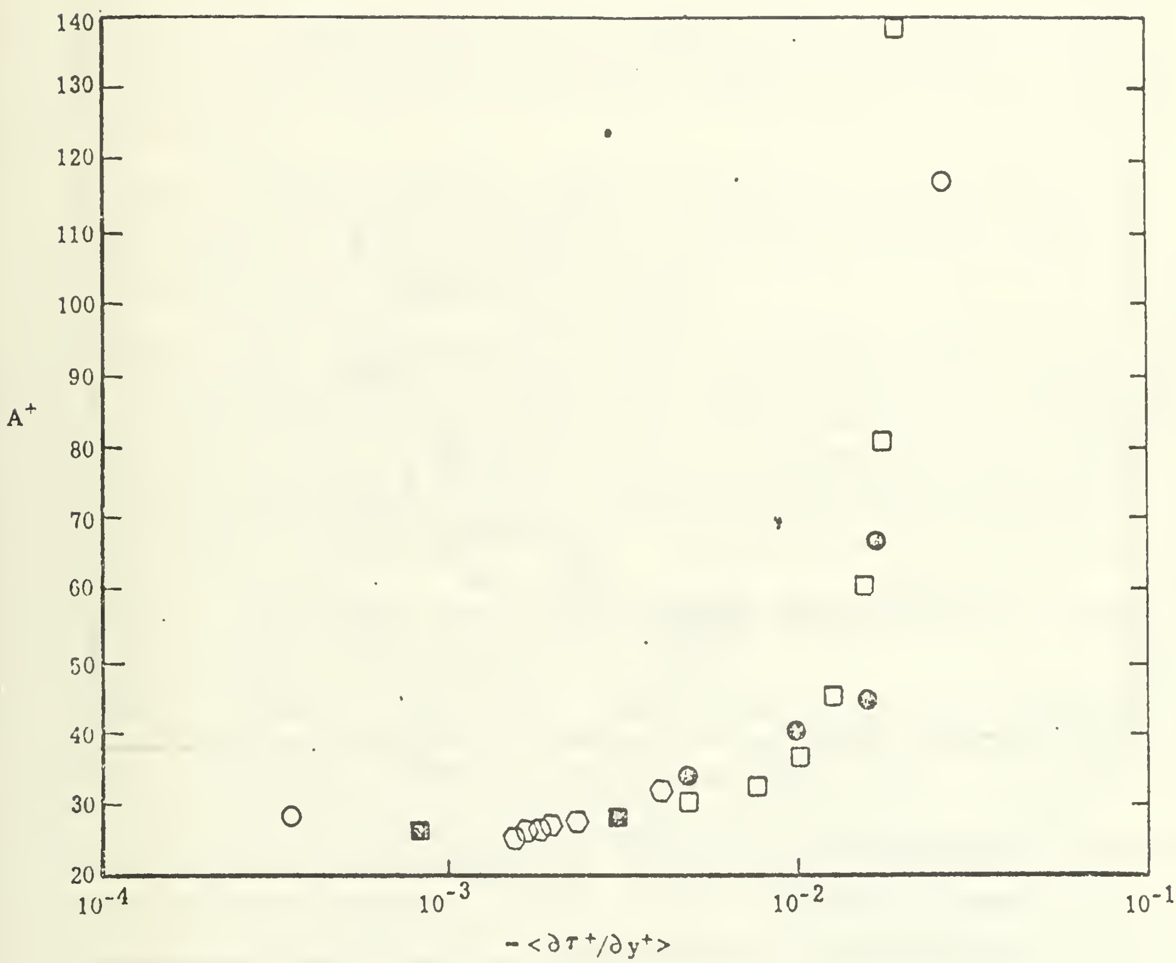


Figure 14. The Viscous Length Scale A^+ for Two-Dimensional Flows. Two-dimensional wall jet: ⬡ Bradshaw and Gee (1962). Fully-developed channel flow: □ Patel and Head (1969); ■ Laufer (1950). Two-dimensional boundary layers: ○ Bardi Narayanan and Ramjee (1969); ● Julian et. al. (1970).

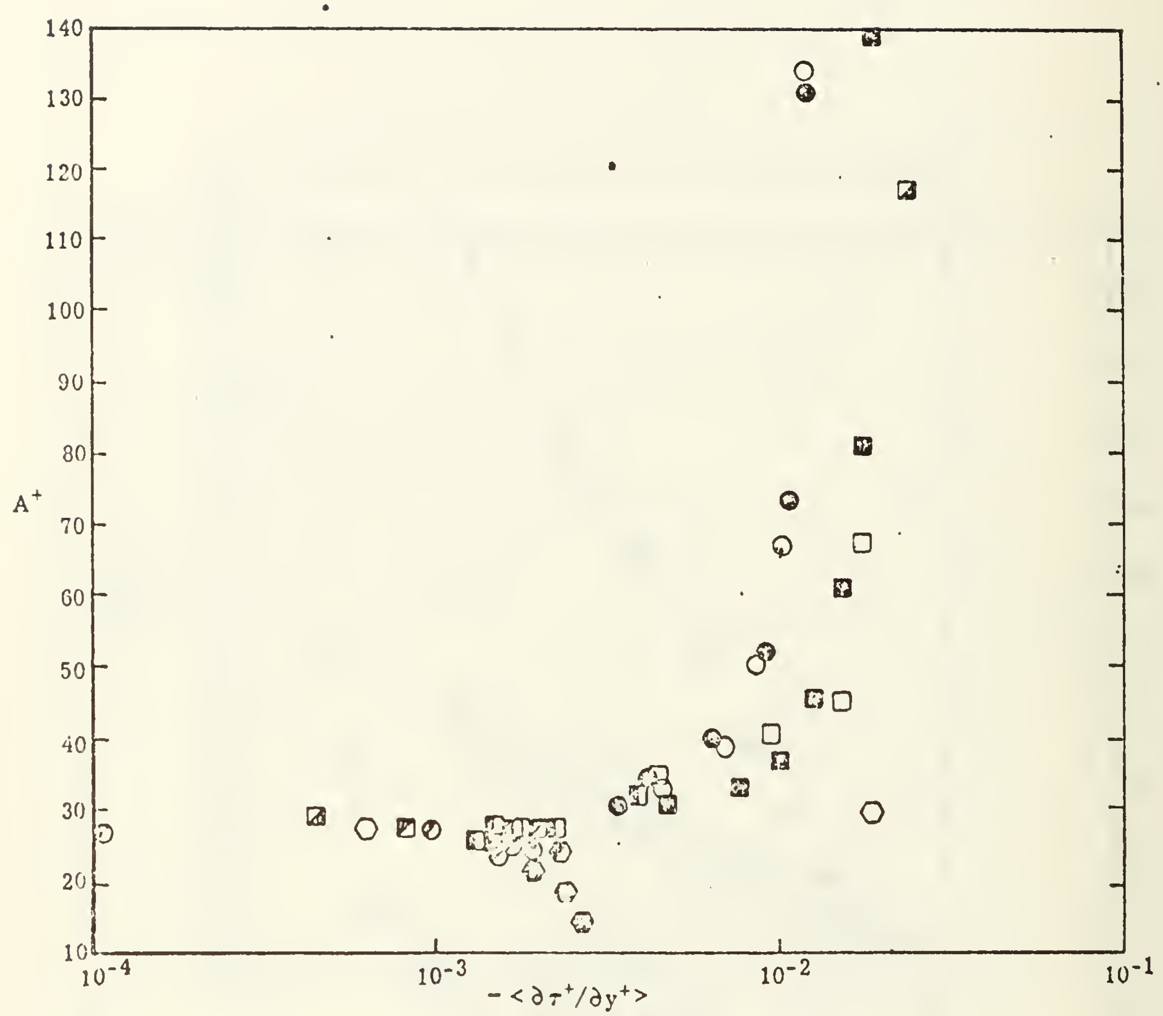


Figure 15. The Viscous Length Scale A^+ for Two-Dimensional and Axis-. Symmetric Flows.

Axis-symmetric flows on concave surfaces. Fully-developed pipe flow: ○ Patel and Head (1969); ○ Patel and Head (1969); ○ Laufer (1964).

Two-dimensional flows. Two-dimensional wall jet: □ Bradshaw and Gee (1962). Fully-developed channel flow: □ Patel and Head (1969); □ Laufer (1950). Two-dimensional boundary layers: □ Julian et. al. (1970); □ Bardi Narayanan and Ramjee (1969).

Axis-symmetric flows on convex surfaces. Axis-symmetric wall jet: ⬡ Starr and Sparrow (1967). Fully-developed annular flow: ⬡ Lawn (1968). Axis-symmetric boundary layer: ⬡ Cebeci (1968).

DISCUSSION

(Herring) First of all, in your thesis, you have used a combination of flux-gradient hypothesis and a turbulent-structural hypothesis; and I notice that having worked with Bradshaw, he has managed to purge the flux gradient.

(Huffman) I abandoned that on the basis of the recommendation of the "Stanford Report" that says, we have enough good methods around and don't need to develop any new ones, rather one should extend existing ones.

(Herring) Another question I have is, in your turbulent energy equation, this is the first series of things that you were talking about, there is a viscous-diffusion term. Is this the term that you were talking about having neglected?

(Huffman) Yes, there is some question as to whether that term should be in or should not be in. This is the equation that Jim is referring to. We should have a term like this in this equation, a ($\nu\ \partial^2\tau/\partial y^2$). There really isn't a good reason for not including that term, aside from the fact that, if we essentially delete that term, we know that the dissipation equals production. We essentially regenerate the mixing-length results.

(Herring) But close to the wall, that term is not negligible; and you are working close to the wall.

(Huffman) That is right; that is not correct. The reason that it is being deleted is essentially that this particular formulation for L is valid under those conditions. In other words, we know that the van Driest modification is correct, provided that you neglect this term. Now, if we included this viscous term, the corresponding term in this equation ($\nu\ \partial^2\tau/\partial y^2$), this particular formulation for L would have to be modified. It was deemed that it wasn't worth that since this is an empirical equation. If we can argue that the diffusion term takes this character, we can certainly argue that we can generate an L function that will essentially simulate the viscous-transport term.

(Herring) That is quite true. It is just a shame to throw out one of the very few terms that you don't have to model.

(Huffman) That is correct. That was my argument exactly. Why throw away something that we know? But I think from a practical point of view, and I've found this in doing the first preliminary calculations with the model, the general solution that we derived does have that term in it. We ran through some preliminary calculations, and we essentially found that we got much better results when we knocked that term out than we did with that term. And I attributed that to the fact that we had L incorrectly formulated if we considered that term. So I think it is possible to put the term back in and reformulate L; but from a practical point of view, I will have to proceed along the path that I have.

(Herring) One last thing if you don't mind, in the second series that you were discussing, the wall hypothesis, I notice that you've gone back to the flux gradient there. How do you account for that?

(Huffman) No, I haven't gone back to flux gradient, Jim. I have gone back to the flux gradient, in that I have assumed that in this reverse-transition region, we've got an equality between production and dissipation. And this is essentially the criterion that was originally instituted to determine whether or not the flow was fully turbulent, or was in the process of changing from turbulent to laminar. So, in keeping with that assumption, this equation reduces simply to the mixing-length equation; and I simply plug it back into this relationship and assume inner-layer similarity. Then in essence, I have done that. That is in keeping with the original assumption that Bradshaw made, that this does reduce to the mixing length in the inner region.

(Lakshminarayana) In the last slide, you showed the A+ going in the other direction. Is this due to the curvature effect, which you neglected in these equations?

(Huffman) There isn't any curvature effect neglected in the equations that we are solving. In effect, we have included all of the mean-flow curvature effects. That's the point of the argument. Curvature has been accounted for. That is why we don't understand the deviation between the two curves.

(Unidentified) You mean the parameter $(\partial\tau^+/\partial y^+)$ has the same physical significance in both of those cases?

(Huffman) Well, the equations account for curvature in both cases. The physical significance of this parameter is uncertain, in that we don't really understand why the viscous sublayer thickness would be a function of the shear stress gradient, aside from the fact that all previous investigations show that it was a function of the pressure gradient, the imposed external pressure gradient, or the presence of either blowing at the wall or suction at the wall. So, if you examine the equation for shear stress gradient, you see it contains a pressure gradient term and a v-wall term; and that is certainly a lot more appropriate to correlate over the inner layer than either of these parameters are. This parameter is an external flow parameter. What can it have to do with the viscous sublayer? It is way out here at the edge of the layer. Granted that v-wall is an inner parameter, but from examining this equation, and from the physics of assuming that the parameter that is related to the wall-like region should be determined from the wall-like region, you can conclude that $(\partial\tau^+/\partial y^+)$ would be a better correlation parameter than either $\pi+$ or B+. And that is how we arrive at that. What we have shown essentially with the two-dimensional results is that that is, in fact, true. It is a better parameter, and it does, in fact, correlate all this information. But what we can't explain is why there is a geometric effect. Admittedly, this is the first time to my knowledge that anybody has gone through all of these different geometries systematically. There isn't much data, and we are making a conclusion that there is a substantial difference on three data points. But I have rechecked the three data points quite a few times, and the data looks good. This is data from Starr and Sparrow, and their information is usually pretty good.

(Mikolajczak) As I listened to the boundary layer discussion which we have just heard, I felt that we were getting away from the real flow in the machine. I would like to discuss the wake transport through blade rows, which was referred to by Dr. Lakshminarayana. Some of the results have already been published (see Ref. 2). The discussion has a bearing on the problem of non-steady flows and blade-row interference effects.

It is interesting to see how the problem arose. We found that the measured stagnation temperature downstream of a compressor stator had strong gapwise non-uniformity, particularly when the rotor ahead of this stator was operating at a high relative Mach number (Figure 1). How was this possible? If you consider the models we have of rotor flow, we usually time-average the periodic wakes, which implies an assumption of axisymmetric flow. That leads to the conclusion that the stagnation temperature, on a time-averaged basis, is constant downstream of the rotor. Since there is no energy transfer across the stator, the axisymmetric approach also requires that the time-averaged stagnation temperature must be constant circumferentially at the stator exit. However, measurements show that there is at least eight to ten degrees gapwise variation at the stator exit. This variation indicates the hazard of using a single probe to measure the temperature rise, and hence the efficiency, of a compressor stage at a spanwise location.

In order to understand the flow, consider Figure 2. The rotor is moving to the left, with a wake having a velocity defect relative to the core flow in the relative frame of reference. If the wheel speed is added to the wake velocities and to the core velocity, we see that the rotor wake fluid has a slip velocity toward the pressure side of the stator. Thus, as the rotor wakes pass through the stator, relative to the main core flow, they are transported like a jet toward the the pressure surface of the stator. The presence of the blade pressure surface will interrupt this transport, with the result that the rotor wakes will be collected by the pressure side of the stator blades and the rotor wake fluid will tend to appear in the stator wakes. Since the rotor wakes usually have had more work done on them than in the core flow, the stagnation temperature of the wake fluid is higher than that of the core flow. This means that the hotter wake fluid is transported to the stator pressure surface and appears as a higher temperature region at the stator trailing edge. The model derived for this flow situation (Reference 2) uses the Silverstein wake (Reference 3), and with some simplifying assumptions relates the temperature excess in the stator wake to the profile loss associated with the rotor wake. Thus, the predicted stator exit temperature profile (Figure 3) and typical measured profile (Figure 1) show that near the suction side of the stator, the temperature is lowest; midgap the temperature is approximately that associated with the rotor core flow; and at the pressure side of the stator, the temperature is highest.

Work by Jack Kerrebrock at MIT using rotating blade rows on a water table has since clearly shown this process. In fact, he showed that as the fluid piled-up on the stator pressure surface, there was a tendency for it to curl up on the upstream side of the "jet" created by rotor wake fluid. The flow pattern looked like a stagnation-point flow imposed on the mean flow moving periodically through the blades. This behavior of the flow immediately raises the question of how does the boundary layer behave? It is not a simple boundary layer problem any more.

A comparison of this theory with compressor data was made.

(Unidentified) How far downstream is that measured from your stator exit?

(Mikolajczak) Temperatures shown in Figure 1 were measured about half a chord to one chord downstream of the stator. Figure 4 shows a comparison between prediction and data from a transonic stage. Theory indicates that along a compressor operating line, where the flow angles and loading are nearly constant, the excess stagnation temperature at the stator pressure surface should be proportional to (tangential Mach number, $M_T)^2$. An operating line passing through the peak efficiency points of the compressor speed lines was selected. The data presented here were obtained at the stator mid-span location. As predicted, the temperature excess was proportional to M_T^2.

We made another comparison. The wake-transport model relates excess temperature at the exit of the stator to the profile loss of the rotor. Starting from the temperature measurements, we calculated the rotor profile losses. The results fell within the NASA correlations and also agreed with losses we calculated using the total pressure measurements across the rotor and an estimated shock loss.

(Lakshminarayana) Do you have any correlation for the wake-chopping effect?

(Mikolajczak) If you are referring to losses associated with wake chopping, we have no correlation. We have concentrated on the temperature problem and related it to the rotor profile losses. We have said nothing about the stator losses. This is probably a fruitful area for further research. If you consider Figure 2 again,you have to ask yourself; is the stator lift affected by the wake transport? The rolling up of a vortex on the pressure surface suggests that there is a vorticity being shed from somewhere near the mid-chord and not necessarily from the trailing edge. This raises the question of how valid is the cascade approach for other than the first row of a compressor? One has to tread very warily and have some rotating-data background for confidence. An interesting question arises in a multistage supersonic compressor where a discontinuity from the first stator enters the second rotor, which has supersonic relative velocity. How do the rotor shocks interact with this discontinuity?

At this point, I want to re-emphasize the need for flow visualization inside a rotor if we are to make progress in understanding flows in our compressors.

The flow in a turbomachine is obviously non-steady. We have to break away from the present-day uniform, axisymmetric flow approach and look at the actual flow. For example, we have to consider what happens if at one extreme we have a wide rotor wake compared to the stator spacing; and at the other extreme we have a large number of thin rotor wakes inside a stator passage. What is the magnitude of the unsteady effect, and at what reduced frequency will it become important? How much further can we load up the blade by making use of this unsteady flow? These are the kinds of problems, I feel, that we ought to address ourselves to.

(Vavra) What about this question of cascade tests and rotating tests which you talked about this morning?

(Mikolajczak) You have to take some precautions when doing this. If one is looking, for example, at a single-stage fan without inlet guide vanes, the problem does not exist in the rotor. There probably we are safe. Currently, NASA is sponsoring a two-stage fan program in which we will be looking at the interactions very carefully.

(Lakshminarayana) Are you trying to measure this flow field inside the rotor?

(Mikolajczak) As everybody else, we are getting on the verge of taking measurements inside rotors.

(Lakshminarayana) What device are you using, the hot wire?

(Mikolajczak) In transonic rig work I discussed, we used hot wires behind the rotor and behind the stator, but not inside the passages.

We also visualized the flow in water. Jack Kerrebrock ran a very nice experiment at MIT while we were developing the theory. He introduced helium into a rotor wake of a low-speed compressor, and observed the helium concentration in the stator wakes. Results confirmed the validity of the transport model.

(Bullock) We have observed that both the total pressure and total temperature have circumferential gradients. Except for the stator wakes, however, the entropy stays reasonably constant. This observation suggests that the pressure field of a stator causes the flow through rotors to have circumferential variations over and above those just discussed.

(Mikolajczak) Yes. The magnitude of the interaction you mention depends on the Mach number. It is particularly severe when the Mach number is near unity. This interaction was graphically illustrated by Professor Vavra's turbine work, where he could not get stable operation with the thick airfoil design and close spacing between stationary and rotating row.

(Smith) I am not sure if I am just repeating what you just said or not, so you can tell me after I have said it. I would like to raise the possibility that the kind of interaction that we have been hearing about might not be an added loss mechanism, but might, in fact, sometimes act to prevent loss. There is the possibility, at least, that the wake that is shed from an upstream blade row can be made more uniform or attenuated by a reversible process, rather than the irreversible process of dissipation, because the following blade row, which is moving relative to the one that shed the wake, will add more energy to the wake than to the free stream. So there is the possibility that some of the entropy rise that would otherwise occur as the wake dissipates will not occur in such a scheme.

(Mikolajczak) This is quite possible. Another mechanism for a possible loss reduction is the boundary layer flow on the suction surface. Here we usually tend to consider the boundary layer to be less stable and responsible for more losses than the boundary layer on the pressure surface. In the wake-transport model the suction surface boundary layer is sucked away as it develops. Does the impingement of the wake fluid on the pressure surface offset this benefit? I do not know.

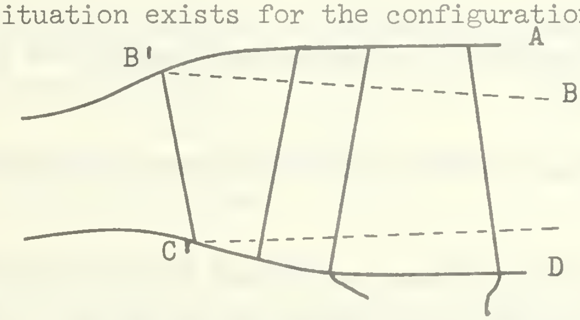
(Smith) We have been aware for some time of the circumferential temperature variations at stator discharge that your wake-transport model so nicely explains, and we always measure temperature behind stators with rakes that sample the whole stator passage. For most cases the wake-transport model probably provides the whole story. But if you consider cases where the rotor and stator blade rows are close together, so that the static pressure fields from each can interact with the other, then there are other mechanisms at work. These may delay separation in an upstream blade row, for example, by lowering the back pressure in the wake. Or possibly the non-uniform work addition by a rotor because of the stator behind it might help to delay separation on that stator. It happens that the rotor adds more total pressure to the flow that goes over the suction side of the stator than average; and hence, it is like a boundary-layer-energization kind of phenomenon. This can play a role too.

(Mikolajczak) The model I described will predict changes in stage efficiency as a function of rotor and stator spacing. The change is, however, only significant for very close spacings; and the direction of the change depends on the design velocity triangles. I do not claim that the model gives all the answers. There is a shortcoming which would merit further work. We have used the wake model of Silverstein, which does not satisfy the conservation of momentum. A better correlation or description of wakes from compressor airfoils, including separated flow, would be very valuable.

(Bullock) This is a plea for the support of effort that may eventually yield both a working model of secondary flows and the understanding of the penalties they impose on turbomachinery. Two significant observations are made to support this plea; both are concerned with turbines.

1. When the hub and casing of a turbine are concentric cylinders, the total losses created in a turbine are, with few exceptions, very accurately predicted by NASA correlations. Secondary flow redistributes the losses, and concentrates them in pockets. They do not induce significant further losses.

A different situation exists for the configuration sketched below.



The overall losses are greater than those predicted by the NASA correlations; in particular, the losses between A and B, and C and D are higher than one should expect. Dust and oil patterns suggest that strong secondary flows begin at B' and C', and a sizable corner vortex is created almost immediately downstream of B' and C'. The angular momentum of these vortices has little or no radial component, and there is not enough torque available in the flow to establish one. Moreover, as the vortex moves downstream, its diameter of influence grows.

The rest of the flow has to move around this vortex to follow the inner and outer contours - almost as if the vortex pencil were solid. Extra losses are suffered, which seem to be a function of the distances AB and CD.

2. Many visual observations have probed the existence of a radial secondary flow from casing to hub along the trailing edges of turbine nozzles. Little or no evidence has been published, however, to indicate any adverse influences of this flow. Recent observations suggest, however, that the trailing edge flow can become alarmingly significant when the following situations occur simultaneously:

a. Supersonic flows at nozzle exit at the hub

b. Low aspect ratios

c. Uncommonly large ratios of trailing-edge thickness to chord length. (Notice that these are precisely identified with low-flow, high-work and high-temperature turbines.)

The losses appear near the hub of the turbine and may account for 20 to 30 percent of the losses generated. One may surmise that the flow near the trailing edge experiences an over-expansion followed by a shock. The trailing-edge losses are augmented once a broad path is available to transfer the boundary layer to the hub. So much boundary layer material is thus brought to the suction surface at the hub that local flow is completely confused, and order is established only after lots of entropy is created.

The mechanisms suggested are speculative, but the effects are real. A little illumination of these and similar events would save a lot of ad hoc development effort.

(Sells) Bob, when you say thick, what do you mean in terms of some dimensionless ratio?

(Bullock) I don't recall that number, but it was thick enough to put a hole through it for cooling air. The trailing edge thickness would thus be between 10 and 15 percent of the maximum thickness.

(Houchens) Did you happen to try that with some cooling flow being ejected? I mean, that probably changes the character of that particular phenomenon.

(Bullock) Ejecting cooling flow certainly changes the character of the wake. In an actual engine, however, we must contend with a new variable, the ratio of the cooling air temperature to nozzle inlet temperature. The radial migrations of the wake, and their effect on the flow, must be a function of this variable -- I am not prepared to give a blow-by-blow account of the physics; I submit these observations only to indicate a problem that has not been discussed in the past. It is an important problem, particularly to small turbomachinery.

(Sells) I would just like to ask one more thing or make one more statement about that. As far as I know from all the data that we have, the loss from trailing-edge thickness is a continuous thing. You are talking about flow coming down the trailing edge, but this doesn't seem to be a discontinuous function that when you get to a certain thickness it disappears. It is not an additional loss. It has to be part of the loss system due the trailing edge, because a curve of loss is quite a smooth function, that is, in the absence of any cooling flows. That is a different story entirely.

(Bullock) The hardware tested had no cooling holes, but the trailing edge was big enough so that large enough holes could be put in it to accommodate a conventional amount of cooling air. The explanation I offer to myself is that the trailing edge is a ladder which passes low-energy air from the casing to the hub. When enough flow is thus passed to the hub, the high-Mach-number flow there just gives up and separates.

(Yampolsky) Bob, does the cooling flow come out in the same direction as the through flow?

(Bullock) When the cooling flow was used, it came out in the same direction as the through flow.

(Lakshminarayana) Mr. McBride, you had some comment on radial body forces.

(McBride) Yes, I was asking if you wanted to comment on the possibility of using radial body forces to control secondary flow.

(Lakshminarayana) I don't think I have an answer. What happens to the secondary flow and the leakage flow when you have a body force due to blade twist in the radial direction? More research is needed in this direction.

REFERENCES

(A. A. MIKOLAJCZAK)

1. Mikolajczak, A. A. etal, "Comparison of Performance of Supersonic Blading in Cascade and in Compressor Rotors". Journal of Engineering for Power, Trans. ASME, January, 1971.

2. Kerrebrock, J. L. and Mikolajczak, A. A., "Intra-Stator Transport of Rotor Wakes and its Effect on Compressor Performance", Journal of Engineering for Power, Trans. ASME, January, 1971.

3. Silverstein, A., Katzoff, S., and Bullevant, W. K., "Downwash and Wake Behind Plain and Flapped Airfoils", NACA Rep 651, 1939.

4. Carta, F. O., "Coupled Blade-Disc-Shroud Flutter Instabilities in Turbojet Engine Rotors", ASME paper 66 WA/GT-6

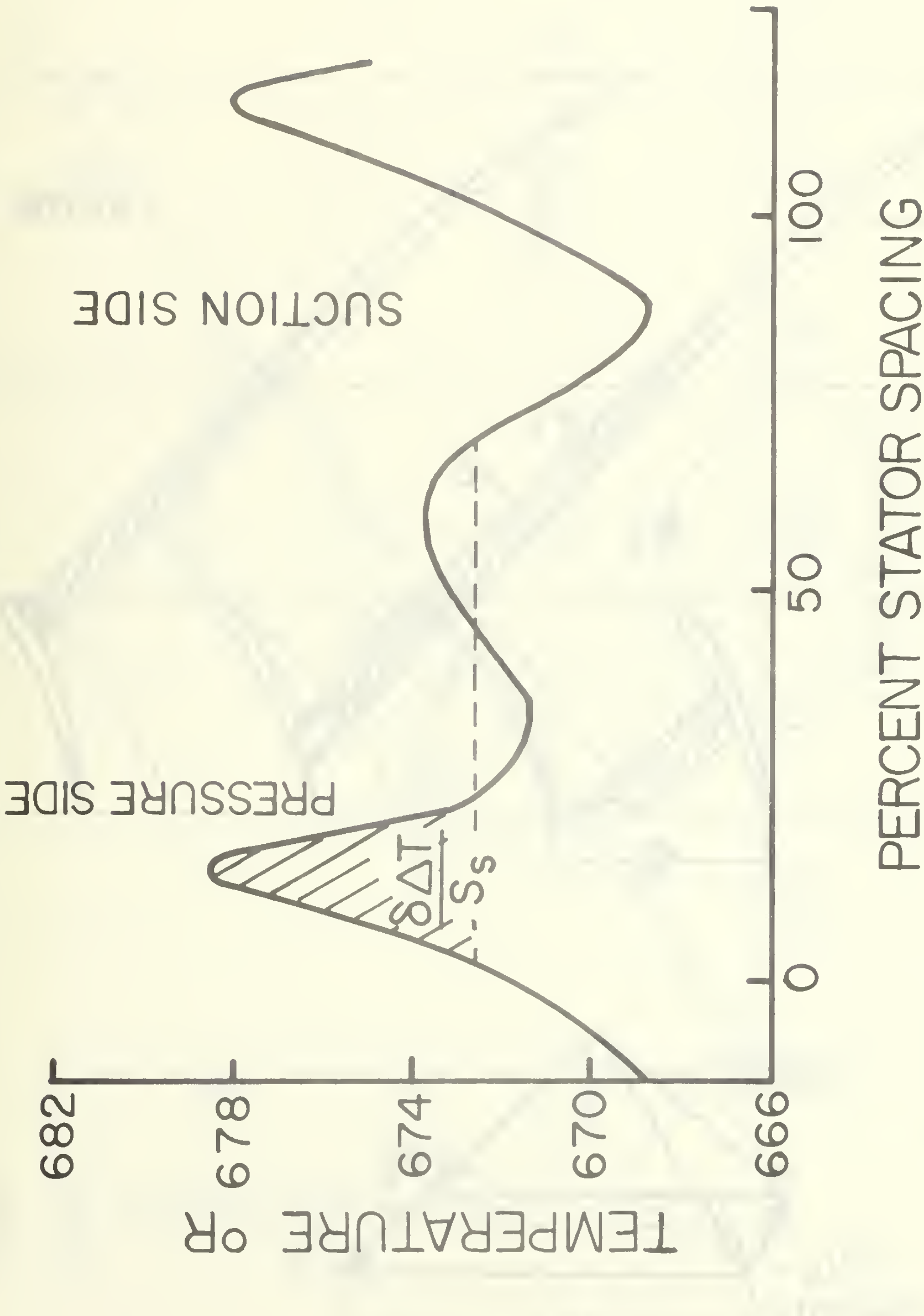

PRESSURE SIDE
SUCTION SIDE
$\frac{\delta \Delta T}{S_s}$
682
678
674
670
666
TEMPERATURE °R
0
50
100
PERCENT STATOR SPACING

Fig. I

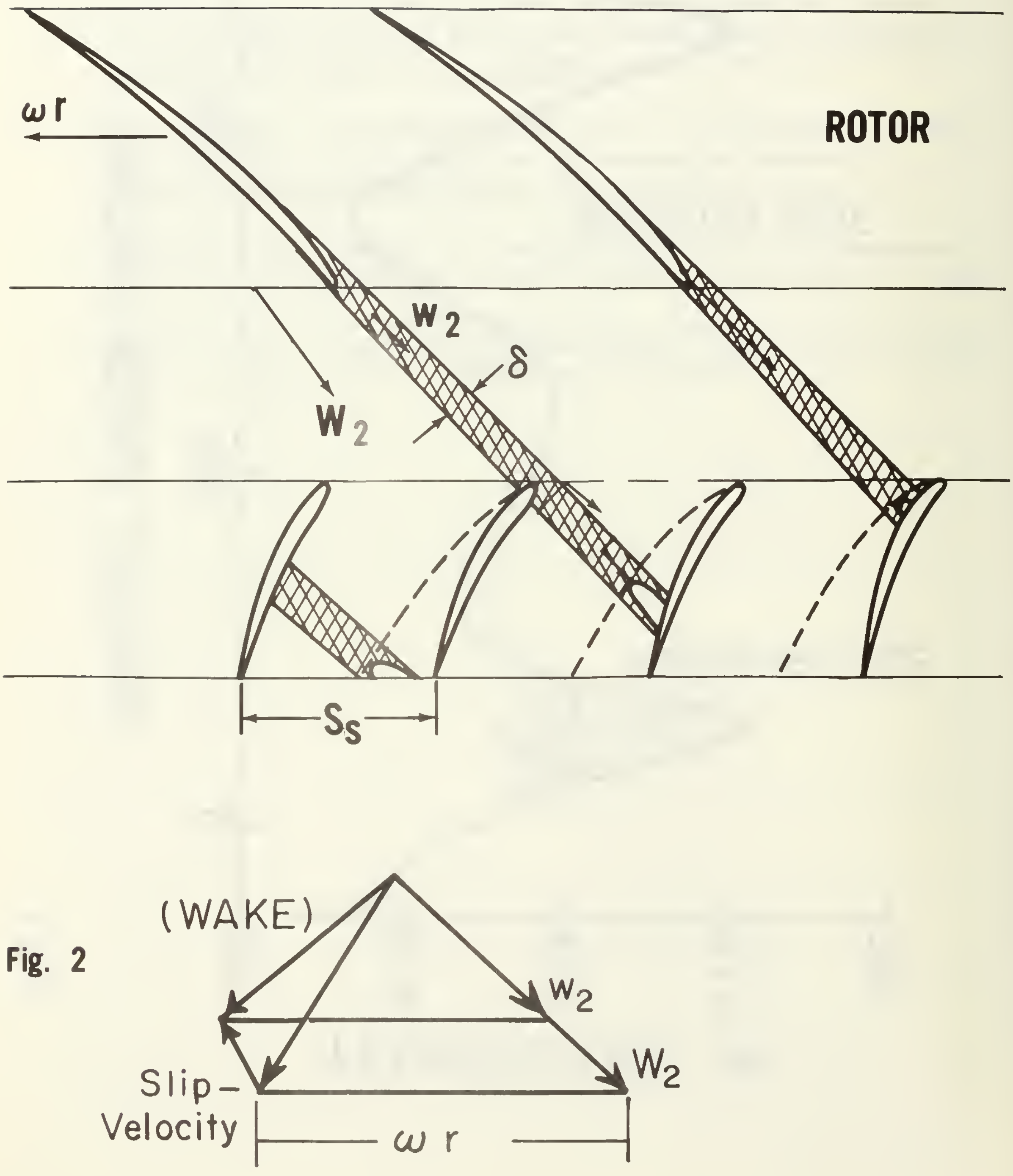


Fig. 2

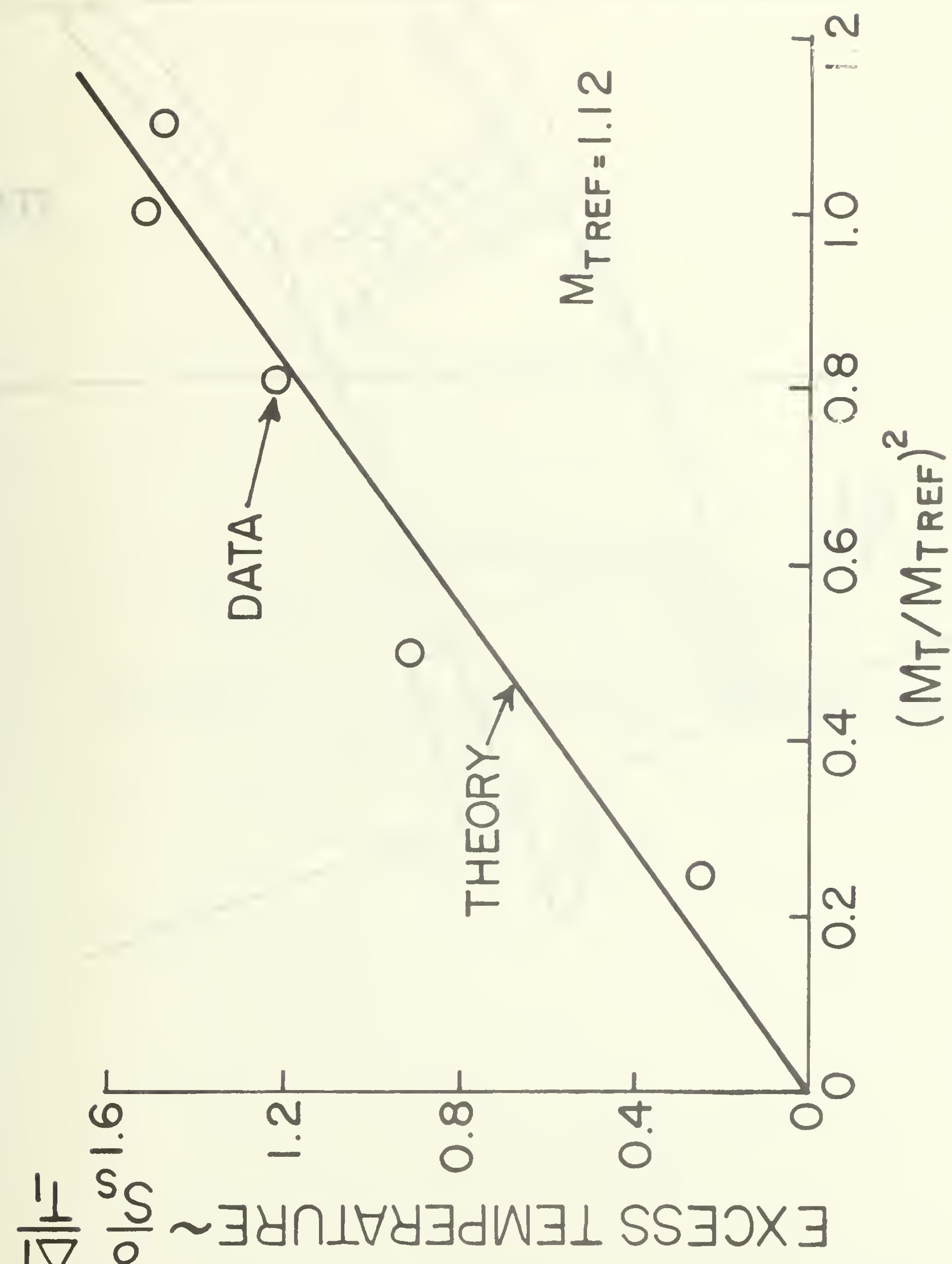

EXCESS TEMPERATURE ~ $\frac{S}{\delta_s}\frac{\Delta T}{T}$
0
0.4
0.8
1.2
1.6
0
0.2
0.4
0.6
0.8
1.0
1.2
DATA
THEORY
$M_{TREF} = 1.12$
$(M_T/M_{TREF})^2$

Fig. 3

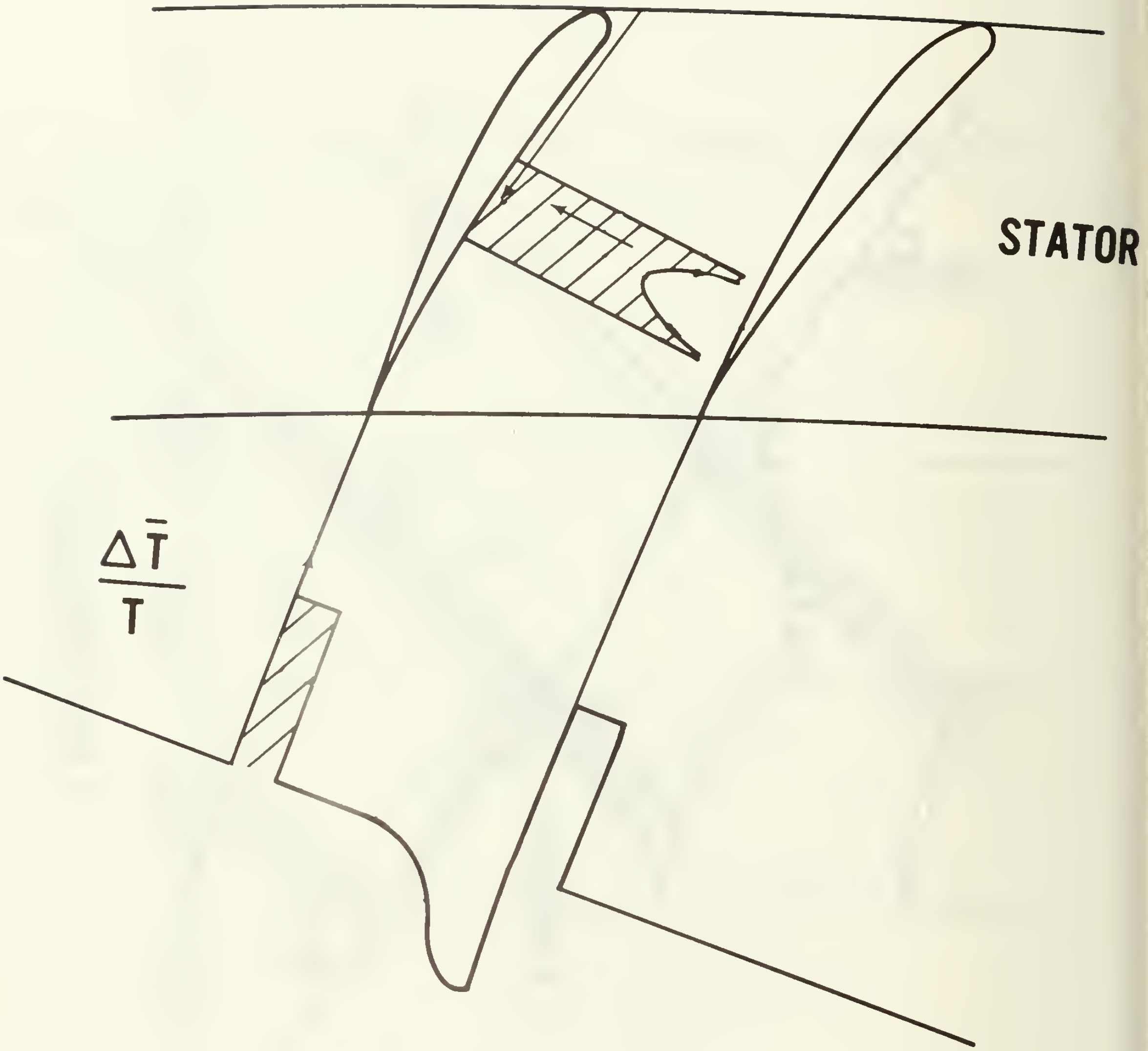


Fig. 4

LOSS EVALUATION METHODS

Discussion Leader: Prof. B. Lakshminarayana

Presentation: Loss Correlations and Off-Design Performance Predictions

by

Prof. G. K. Serovy

Loss Correlations and Off-Design Performance

by
G. K. Serovy

Alex Mikolajczak said something yesterday. I think maybe he was joking, but I didn't take it that way. He said, "Maybe you ought to tell us what you do when you design an axial-flow compressor, and then we can pick on you a little bit". I would like to have that done. Another comment was made here today about something that really is true. I don't think many people design axial-flow compressors from scratch. They scale, and they fiddle, and they twist, and they cut off stages a lot of times. So, I suppose the question is what do you do when you have to design one from scratch. This also leads right into the subject of performance calculation, which is what I am supposed to be here to talk about.

There are four steps in axial-flow compressor design. This is when you start from scratch. The first one of these is what I call preliminary design. I really consider this to be the hardest one of all in many ways. Because this is the one where a man says to you, I need a compressor that produces a certain pressure ratio at a certain flow rate. It will rotate this fast. I would like to have you get the best possible efficiency that you can produce for me. Come back when you've got the answer. This is the step where you determine the size of the machine, how much diameter, hub-tip ratio at the inlet, number of stages; and really at this time you make some very important decisions. When I have students in class I always say, "You had better be right when you do this, because this is the place where you have to fix in a preliminary way such things as aspect ratio of your stages, and as I said, hub-tip ratio, tip diameter, axial-inlet-Mach-number level, and things like that, that significantly give you the business later on, if you make the wrong guesses at this time". The second step in the design process is the one that really has been talked about here most of the day. This is the calculation of annulus shape, and velocity and property distributions in the passage. And I should point out that in most of the methods that have been described here, or have been alluded to in the discussion, you really have to make a guess at this point about how the blades are going to look and how they will affect the flow as you make this calculation. And remember, you are doing this before anybody has selected any blades. You have no idea what they really will look like, because you haven't gone to cascade data or whatever you use in the process of finding out what the thicknesses are going to be. But you have to have some of that information in order to get into this annulus shape calculation. The third step, of course, is selecting the best possible blade sections to work with these velocity diagrams that you have determined in the second phase. The fourth operation is that somewhere along the line, you have to predict what the off-design performance of this machine is going to be. Whether it is one stage or four or six or whatever, you have to make a guess at how it is going to perform off the design point. Can you flick on some of those slides? (Figure 1) I used this a long time ago to show what the flow model might be for step two in the calculation. This is the determination of annular shape, and of the velocity and property distributions at various positions in the annulus. The models that you find nowadays are remarkably similar, when you investigate what the various companies do. And I try to investigate what the various companies do, because I operate on my own; and I like to read what General Electric does, and what Pratt Whitney does, and what Bob Bullock does, if I can find out. The models are really all steady, axisymmetric flow. Now, maybe you guys do something else that you don't tell us about when you write reports, even for your sponsors like NASA. If you know this that you don't report on, you have got it remarkably well hidden, because I notice that when the compressors are built and tested they still don't work very well sometimes, even at the design point. It is amazing how much better they are doing now; and I don't want you to think I'm insulting these fine people, because I really feel completely inferior to them in every way. I know that they are all smarter than I am, and I want them to know that I believe that. In calculating here, they use axisymmetric, steady flow models. They all use some kind of radial-equilibrium approximation to determine the radial distribution of fluid properties. They all go back to some very crude loss and deviation angle model, that I will refer to. These are very important, by the way, but they all use these things. They all have lots of terms in the radial-equilibrium approximation. They also have a little thing called continuity that they have to satisfy at every one of their calculation stations. There are a number of these systems that now work inside the blade rows. You know, like 15 or 20 years ago when we had the NACA, we didn't think too much about working inside blade rows. Now the General Electric CAFD program does this. I'm sure that Pratt and Whitney does this also; I know they do because you can see it in their performance. So, where I show calculation stations between blade rows, there may be some calculation stations in the blade row. That really throws some hookers into the calculation system, because then you have to know a lot more things about loss and about the way the flow goes than you do if you use those between-row stations. No matter where your stations are located, if you use the so-called radial-equilibrium solution, you have to start out by knowing some things. If you are designing, you have to set a total enthapy gradient and rV_θ distribution. This is along the radius now, at every one of the stations that you work with. Nobody in his right mind would try to design a compressor now without using what he thinks are correct loss distributions, so everyone has entropy gradient terms in it. In the old days, it was just assume an efficiency for the stage, and then don't worry about what the entropy gradients were. In fact, I just remembered the late Jim Hatch, I always like to give that fellow due credit for being one of the first guys - maybe Bullock was prodding him - but he was one of the first guys to recognize that this really was an important factor. I think that industrial people were also on to this at that time, but Jim was the guy who put it in a report and he deserves a lot of credit. Everybody also has to determine the effects of stream-surface slope and curvature. You have these axisymmetric stream surfaces; and if you use those, there is a very important effect in an axial-flow compressor of the shape of these surfaces on the axial-velocity distribution, which is going to be the end product of all of this. These stream-surface slope and curvature terms are found by an iterative process. First, you have to make some preliminary guess or some preliminary calculation. The problem of establishing or actually determining these curvatures and slopes is not an easy one, especially if you are setting up a design system for the first time. I notice that General Electric when they use their intra-blade solutions, they tell us that this helps to stabilize the situation with respect to the determination of curvature and slope. Maybe Roy Smith would be willing to make a comment on that later. They indicate that because they work within blade rows and have more stations along the stream surface, that this feature helps them to do a better job of getting what they think are more correct curvatures and slopes to include in the distribution calculations. Way back when, in the radial-equilibrium equation development, we decided that we could do without local shear stress terms. These are the things that our boundary layer friends have been talking about all along here I'm afraid, and I'm also afraid that we are going to have to do something about those one of these times, particularly near the walls. We had a guy named Fujii, from Japan, with us a year or so ago, who was an exponent of trying to do something about this. What happens in the case of part-span dampers, and what happens near the walls when you have a disturbance in the flow? Maybe we shouldn't forget about this completely. There are some people in Germany who think also that we shouldn't neglect these things. Blade forces, of course, if you are inside the rows, you have got to include these blade force components. This is a significant part of the CAFD system and Pratt and Whitney systems, I am sure.

The end product of a radial-equilibrium equation solution at any one of these stations is an axial-velocity distribution that has to satisfy continuity. If it doesn't, you go back and start all over again, and recalculate based on another axial-velocity guess, in order to get the right mass flow at each station. It is a really tough proposition; and remember, you have to do this at the design point. As I said a while ago, the man, when he walks into the place and says I would like to have you design a compressor from scratch, expects you to do step number one, the preliminary design, sizing, and all of that. Then he wants you to do the second step, the velocity and property distributions; and he wants you to do through all this affair. He doesn't want to spend any money to do this, of course. When he comes to me, he wants me to have the programs all developed. In companies, this is not much of a problem, but for us it is. For any of you guys who are starting from scratch, you know the NASA bought such a series of programs; and the Allison Division very competently set up a series of computer programs, which can be used for what you might call phase one and phase two of this design process. They have a good program for preliminary design; we have used that. We have used the one that is the second phase program; you have two in that package. But the second phase program is a pretty good one. It has some faults, such as when you get loss distributions, you only have three points along the span in your loss curve; and that creates some problems, which I am sure you have solved maybe for your own purposes. You guys certainly have it solved. You can buy the package of programs which includes, besides all this, an off-design performance program from this computer library called COSMIC down in Georgia. They will sell you the program package for $300 or so, and you get a big package of books and a deck of tape. And I want to report to you that when you go to put this on your own computer, it costs you about $4000 in time and untold grief in order to get it on your system. I know that because we just finished doing this; it is quite difficult. Don't think that you are off the hook when you pay that $300. I don't know what is going to happen to us if we ever have to do this off-design program of Allison's. It looks very good, they have a lot of things in there. Bob, don't think I'm really giving you the business, because I have the greatest respect for the guys who did that program. They did an excellent job. The only criticism is that I don't want these people around here who sponsor work to think that we have it all done yet. Sometimes even you industrial men just wave your hands and say, we solved that problem years ago. I'm afraid that we have a long way to go, even in the development of some of these programs that somebody called two-dimensional programs. That is very bad; they are rough. We are leading up to the off-design situation.

In theory, when you finish with your design steps, the first three steps, you have picked your blades; and you build the thing. You put it in the test facility, and you run it. It operates at exactly this point, and it produces 94% efficiency. But, of course, it doesn't. Even in cases where good flow models have been used, you get some very funny-looking results. The actual operating point lies out here somewhere or down here somewhere, and then you have to do something about that. So, I am pleading that we don't stop working on these simplified models.

Let's flip through the rest of these slides. We use these axisymmetric stream surfaces. Fagan was talking about really the first step in going toward not using axisymmetric stream surfaces, working with real ones. And I want to reiterate my comment loudly to everybody that this is a feeler going in the right direction. You have to keep sticking your nose out there and getting it cut off very frequently, if we are ever going to get better systems for design. So, don't attack these men too much right now. At present I think though, the only practical systems for design that work with the data we have available are still the steady, axisymmetric systems. I would like a comment from any of you guys who differ with that.

Cascade flow, sure, we have to use that. We still use cascade data, and we use cascade correlations. I am going to try to indicate to you, maybe quickly tomorrow morning, how this is done. Sad to relate, when people pick blading in the third phase of the design process, they still use to predict deviation angles things like Carter's rule, which Wally McBride told me last night is from 1939. But they add to Carter's rule a fudge factor that may apply only along a part of the radius. They use something at the hub, and maybe they don't use correction toward the tip. But cascade flow is still pretty important, and I agree that we can learn some things by testing in both plane and annular cascades. We have to be very careful what we do, but there are some things that can be done. Pratt and Whitney is spending a lot of money, at least, if there isn't anything to be learned. And I think that papers like Alex put out at Brussels, about correlating rotor measurements with cascade tests, show that there is something to be had here.

Talking about this deviation angle business, here is some General Electric work. (Figure 2) They built some rotors for NASA. They had design values of deviation angle across the passage. These are 0.5 hub-tip ratio, axial-flow rotors. We are just working with one rotor, not preceeded by guide vanes, and making measurements downstream. We are comparing measurements for these four rotors with the design values. The design values are the lines, and the data points are the triangles. Down below, I am just showing what the deviation angle adjustment was that was used. This was an item called X that was added to the Carter's rule deviation angle. Carter's rule was kind of fixed up a little bit here. They used an equivalent camber angle in that; and when you try to duplicate their work, you had better read it very carefully. They had a very good operation, a good way to design. But remember, these guys are engineers doing this; and they had to fudge things a little bit sometimes, even before they added to the X factor. Remember, these are the leading companies in this country that we are talking about here.

One more thing, when you do this phase two velocity-diagram determination or property distribution in your steady, axisymmetric calculations, you have got to have losses in there. Remember, you have got to put these losses in before you really know what the blades look like; and so this is quite a problem. This is real old data.(Figure 3) I took this from a report by John Creagh on a transonic stage; and he put in a design value of total pressure loss coefficient, which is the dashed line down below. This was a long time ago, so it wasn't very good. They do a lot better now, but there are still some aspects of this bad situation in existence. This loss distribution was put in in such a way that they tried to maintain a constant total-pressure ratio for all elements along the span of the rotor. In other words, they tried to design so that each element of the rotor would produce the same pressure ratio. They didn't succeed, because, as somebody at NASA told me a while ago, if you design for losses, you surely will get them. The losses that they got were considerably in excess of what John Creagh thought they might be. Old-time loss correlations, not very good, we got very poor results in terms of checking of the actual data. These are data for three weight flow rates. One of these was quite close to design flow rate, and you see it is not too good. The distributions that result from this kind of mistake, if you want to call it a mistake, in terms of axial velocity are kind of bad, because they affect what happens in the stators. And then there is real trouble in subsequent stages.

I want to stop with the slides now, and maybe say a few things tomorrow. I hope I have opened up some subjects for discussion. We have looked today at a lot of interesting phenomena; the intra-stator wake transport, the bleeding down the trailing edge, and others. We get so tied up with these interesting phenomena that often we forget to do something about these design systems that continually cost us thousands of dollars.

I shall deal with a couple of aspects of some of the work that was done with the prediction of performance for NASA. I am sorry to say that this really has nothing to do with axial-flow compressors at the moment because this was the prediction of the performance of some axial-flow pumps. But some of the problems that you run into are obviously the same. We have tried both compressors and pumps in the past, and there is no reason why you can't make the same mistakes and run into the same difficulties with the compressors as with the pumps. We started this a long time ago. This is a typical university research project that moves very slowly, and so there is a long history behind it in time. What we do is to follow what I suppose is a typical procedure for performance predictions. The idea is that we are supposed to not only estimate the performance at the design point for a pump stage, but we are also supposed to estimate performance with some accuracy at off-design flows, possibly at speeds that are other than the design flow. We never got into the cavitation business. We have tried always to predict performance for noncavitating conditions of flow. The solutions or the methods we have used have all been based on the ideas that I talked about yesterday for design. They are all steady, axisymmetric flow situations where we use the radial-equilibrium equation to get the velocity and property distributions across the passage. We work only with stations at the inlet and exit of blade rows. We have worked with individual rows principally, like rotor rows, but we have also tried to work through one stage. We have never been able to be very successful with getting through a many-stage pump for the reasons that I can tell you about in a minute or two.

I guess this is really a confession of just what a hell of a shape we are in in some of these problems, because we have had pretty decent guys working on this through the years, and still we have these difficulties in getting this thing to work out properly. Our computer programs that we use, and obviously you have to do this on a computer, are based on starting, at least the ones we have now, are based on starting at a flow rate near the design flow and at the design speed for the particular configuration. Now, remember we are not designing something now, we are trying to predict performance so we have the complete geometry of the pump. We attempt to start here and to do the best job of estimating what the performance will be for the geometry at the design flow rate of speed. Then, we work from this point towards higher flows, we then work from this point towards lower flows, and we go until the computer stops and tells us we can't get any more solutions. I will say a little bit more about that.

When we start at this point, of course you obviously have to make some initial guesses in order to get a solution here. I won't go into the details of this. We have given some progress reports that are in NASA; but the basis for the program is that after you get a solution near the design point, you move to a flow rate that is very slightly above that and to start the solution at this next situation you use the angle distributions, the flow angle distributions, and the losses that you had here that were the result of the final answer for this point. So you see it is a process of stepping from one flow rate to another, while keeping the machine operating at a constant speed. It is kind of like the testing process that you use when you actually test a pump rotor or a compressor rotor; set the throttle, set the speed, then change the throttle setting, and attempt to measure the performance at different flows. Well we simply move from one flow to another using relatively small increments of flow, and we use the previous solution as the starting point for the iterative process of finding the solution at each successive point. Whenever you make one of these solutions, you use the radial-equilibrium equation as the first or the inside loop of the thing. As I described to you yesterday, you have to satisfy radial equilibrium to get the velocity distribution across the passage. After you have done that, you can see if that velocity distribution satisfies continuity. Then, after that is done you might see if you have shifted your streamline positions or your stream surface positions in the passage from where they were on the previous run. And after all this is done, involving looping in the computer program, you then can readjust the loss distribution and deviation or flow angle distributions in order to get still another guess and start all over again through this affair.

Now, you see this is a very lengthy process or it seems as if it ought to be a very lengthy process for each flow rate. Now, here are some of the things you run into that you don't think about until you try to make an arrangement of this kind. Keep in mind that we are not just trying to get the overall performance, we are not just trying to estimate what this map looks like; what we want are the radial distributions and when we say performance predictions, don't give me any of this stuff of the mean or pitch line solution. I know everybody has got that, and everybody is pretty satisfied with what he can do in terms of performance map predictions. But if you try to predict the detailed flow distributions in the passage, what happens along the radius, this is where these little difficulties come up. We don't have so much trouble in moving up in flow from approximately the design flow rate. One of the big difficulties we had with this thing a few years ago was when we tried to move down in flow from the design point we would get loss distributions -- we didn't have so much trouble with flow angle distribution across the passage down here -- but when we get down in this region, we get loss distributions that would work within the radial-equilibrium equation to give us a failure or an inability to solve the radial-equilibrium conditions. The radial-equilibrium equation, as set up on the blackboard here, comes out to be sort of a quadratic arrangement; and what would happen to us is that we would be computing V-axial, the axial component of velocity at various positions along the radius in the passage, and we would get to some point in the passage where V_z would be computed and it would turn out to be the square root of a minus number.

Now maybe all you guys have encountered this and maybe this is a common experience, but at the time we never did know about it and we still to some extent haven't got an answer to this. All I can say is that it happens to you; and when I say no solution here, I mean that we can't get down into this area where there are stalled elements. You say what's the sense of that anyway, we haven't got axisymmetric flow, we haven't got steady flow then; but in some cases we can't get down to flow rates where NASA, for example, has been able in experiments with similar configurations to get good results. Of course to get not good performance, but to get a velocity profile across the passages, that looks OK. I won't bore you with any more details, because we have a great lot of stuff to hear today, and I won't show any slides to get you educated about all these stupid problems we have had, but take a look at this passout here. I will say a couple of words about this because there are some nice things here for turbomachinery people.

The first un-blank page here (Table 1) says NASA axial-flow pump rotor configuration descriptions. Mel Hartmann back here has been running a long program on axial-flow pumps. A lot of you guys know about this and maybe the compressor people aren't currently interested, but put this in your file because some day this data is going to be extremely valuable and I think will be sought after the next time we have a big fire drill to develop axial-flow pumps. That will be sooner than any of us think.

We have taken all these configurations that you see here and we have worked with the data in order to develop our loss correlations and deviation angle correlations that were used in this program curve. There is a lot more to this than just me talking about performance prediction because I want you to know that this data is around. We have now put it all on tape; we've got it all sifted so that we have the mistakes out of it that might have been made in the original experiments.

Just look at this stuff and see what a neat package of information you have here. You've got, if you look through the data, data on tip-clearance effects in axial-flow pumps; remember, these are all rotors, there is only a stage or two involved here, but there is tip-clearance effect. Notice what he has done here, he has some nine-inch rotors and some five-inch rotors where there is really data on scaling involved. You can get a lot of good stuff out of that. He's got all kinds of hub-tip ratios, of course they are mostly double circular arc blading, but look at the range of D factors at the tip that are involved here. The nice thing about this is that they are involved, and now we have pretty complete information on the design and on the characteristics of these rotors. So you can work with this data in correlation, and as I say, all these guys that have a new idea should take advantage of information of this kind.

The second thing in here is a set of curves that look like this; these are some loss correlations that we have used in a little program we had for design of axial flow pumps. (Figure 4) A guy named Pat Kavanagh and Max Miller, who many of you have heard of as being with Mel a few months ago and now is in school again, wrote a nifty little program for the design of axial-flow pumps; and these are some of the loss correlations they used with this. I was going to say something about this, but I won't go into that at this time. On the third page there is something here I would like to have some comment about, if anybody has any neat new suggestions.

We have to, in our performance predictions, get deviation angles. Now for you boundary layer characters, I heard some snide remarks last night about well what in the heck is this A plus or A or whatever they are talking about, I don't know that. Well maybe it is true that when we talk about deviation angles and D factors, the boundary layer people don't understand what in the heck we are talking about. So, the deviation angle is simply the angle between the tangent to the cascade airfoil meanline at the trailing edge and the actual flow angle. It simply means that the blades in cascade don't give the flow perfect guidance, they don't produce the turning in the flow that they might if there were an infinite number of airfoils in the cascade. And the deviation angle is positive in the direction of less turning than the camber of the airfoil would indicate. In the design business you correlate this deviation angle as measured against various things; you take cascade measurements, and pump measurements and whatever you have and work with this. Now at the present time in this business of pump performance prediction, we are using a deviation angle relationship that looks like this, which gives Carter's rule with one little modification on it. (Figure 5) Max Miller has this idea that we ought to put an exponent on the camber,and the exponent that you see plotted as a vertical axis on this curve is the one that he is trying in the performance prediction program to see just what happens. We got some screwy results in the past from experimentsthat we are attempting to get closer to with the calculations. You see that all this is a plot of this exponent against the difference between the optimum or reference incidence angle and the actual incidence angle that you have at any particular flow rate for the section you are working on. It is the worst form of empiricism I suppose in some ways, but tell me better if you know.

Now, the next sheet shows you some of the kinds of comparisons that we got. (Figure 6) I will have to say what you are looking at here is a comparison between some calculated radial distributions of certain items and measured distributions. The measured distributions were taken by NASA, Mel Hartmann's gang in Cleveland, on one of these configurations that you saw listed back here. This is the configuration 15. You can look at the details of that if you like. Now, what you see plotted here besides the experimental data are computed distributions. The dashed line that you see is the case in which we went through our performance prediction program, the computer program with an input in which the loss distribution was the same as was measured, but the deviation angle that was used was the simple Carter's rule without the gimmick here. You see that the distribution of deviation angle, for example, for the dashed line which is the old Carter's rule, is horrible compared with the measured distribution of deviation angle. The loss coefficient of course is right on in every case because we put in the experimentally measured loss coefficient just as sort of a check run. The axial velocity distributions and the outlet flow angle distribution, you can just see what they look like, but using this exponent as a part of the deviation rule you get quite a bit better deviation angle result, that solid line, than was available with the simple Carter's rule. So, in that respect, we have helped ourselves a little bit.

(Mikolajczak) Was the correlation obtained from more than one rotor?

(Serovy) Yes. It was for numerous rotors from this list. We took Mel's data, or actually his group's data, and we correlated or Max correlated and took several of these configurations, not all, and used them to develop this business right here. It is a fit to a number of the rotors in that list. So it isn't exclusively from this particular rotor that we are trying to calculate for, in fact, configuration 15 was not one of the ones that was used in developing the correlation here. So this is a pretty good check on the deviation angle.

(Katsanis) What exponent do you use for that deviation angle?

(Serovy) The dashed line is just plain Carter's Rule, that is just what you see here without the "b" and the "b" is obtained from the curve.

(McBride) When you laid out basically your double circular arcs how are they laid out? On stream line surfaces? Or tangent planes? Or cylindrical surfaces?

(Serovy) When they were laid out in the design of the pumps, they were laid out along cylindrical surfaces. As we use them in the program, what we do is we have a radial distribution of the metal inlet angle at the inlet to the rotor and a radial distribution of the metal outlet angle, and those are in the input to the program. This is a part of the geometrical input. We assume that even though the stream surfaces don't cut the blade rows at exactly a cylindrical section that you still can deal with the airfoil between here and here as if it were like pretty close to the results.

(McBride) Okay, so you are consistent on your "b", this kind of thing.

(Serovy) Yes, I think reasonably consistent. This is really not so great; what I am really trying to point out to you, I guess, is that you can work for a long time and still have a lot of difficulties with this kind of situation.

(Louis) Was it the same kind of design flow for the design point; all these pumps were designed with the same philosophy, distribution of outlet angles, etc.?

(Serovy) The same philosophy, I guess you could say they used the same design system which was a radial-equilibrium system but as for, say something about that, Mel.

(Crouse) I think they started out using compressor methods for the first designs; but as the data came in we updated, so it isn't entirely the same.

(Serovy) Now that I've had a couple of seconds to stand here and think about it, there really are some major differences like way back in the beginning they were really floundering around. They didn't know how to design pumps; and by golly, what they did was to say let's try what we did with compressors so they used some of the same SP-36 — is that familiar to you? — SP-36 methods and just modified it for constant density flow, and away they went and they built the first few stages, the first stages or so. Then they realized that the deviation angle rules were not so hot in SP-36, at least the corrections that apply for going from cascade to compressor; and so they changed that, they fudged that around. Then they started doing something like they put in loss distributions. At the very beginning, they weren't putting this in; they just assumed rotor efficiency. Then they started using loss distributions in the design process, so there was quite a development of the design philosophy as they went along from configuration to configuration.

We also have messed around with loss correlation a lot, and we are not satisfied that, I am sorry to say, the D factor and D equivalent are here to stay. You boundary layer men, we really need some help in terms of getting parameters that we can use to predict when we are going to get in trouble with diffusion in compressor and pump rows. I am sure you are all well aware of this, but we are still working with this D factor and D equivalent that were developed quite a number of years ago, and with all the advancements that I hope you have been making in boundary layer procedures, I know they have been made, it just keeps sticking me in the back of the head that we ought to have something new coming up. I don't see as much evidence as I'd like of that. Then, what we have done in losses, the partial excuse for this is what they keep telling us why don't you get something done. Again we went back and tried some rather elementary procedures. On one of these pages you see a whole list of equations (Table 2), and these are some of the parameters that we have used in attempting to make loss correlations. We have tried various combinations of items, most of these are reported in detail the way that these were developed by Ted Okiishi; and Ted went back into some of Lieblein's original work and tried to fool around with that until he got some possible correlation parameters especially in terms of (θ/c). These correlations have resulted in what we think is a reasonably optimum next trial. We are going to stop this pretty soon, but maybe a last trial of loss correlations. You see (θ/c) plotted against D equivalent in this curve. This is the one that we have tried to put in in the most recent times. (Figure 7)

See the way this performance prediction program works is that we have a main program, and then we have subprograms for deviation angle and loss. We can put in anything we want, including the experimental values, at the subprogram level. Most of our work recently has been trying to get these various trials into the main program.

The last sheet, and then I will get out of here, is related to trying some of the more recent, this latest correlation (Figure 8). When it says new correlation, it means this correlation right here. Again, we put in experimental deviation angles in all computing cases. But we've tried to show you some of the differences that can occur between using two-dimensional cascade loss predictions and this correlation. When you say two-dimensional cascade loss correlation, we just mean a loss parameter plotted against D equivalent taken from the old NACA two-dimensional cascade work. This is old stuff, but it is a good starting point and again you can see some of the changes that are made.

We are really having trouble with loss correlations. Now, if this happens to you in pumps, you compressor friends, we haven't even mentioned things like adding in shock losses or things like that that have to be done in compressors.

Remember, we are saying that this is sort of a profile loss that you get out of this business, and we don't make any additions for secondary losses. They are supposed to be in here. In effect they are in here because we have different loss correlations used at various positions along the span.

Well, things are tough all over these days. We keep trying and as long as somebody will listen to us, we will see what we can do about both pumps and compressors. I again emit an urgent plea to the research administrators around here; don't stop supporting some of us, force everybody into doing some of this kind of work in developing these design procedures along with studying of the interesting phenomena that we see in these machines.

(Lakshminarayana) Do you have any boundary layer data at the rotor hub and the tip, detailed surveys near the wall?

(Serovy) There is not much of this available. Jim, do you know anything about what is around? There were some surveys taken. You might write to Max Miller. Now Max is the guardian of all NASA data. I will tell you right now, those guys have a bad habit out there. I worked there so I can criticize. They have a bad habit of taking all this new data, and then you come back about four years later and say, gee, can I get some data, I would like to make use of it in some work I am doing and it has gone out to warehouse B-7. You go out there, like at our place, and the rats have eaten it or it has disappeared and you can never find it. So we have got what we think is a complete duplicate set at Iowa State, and I have it locked up in my file cabinet, and Max Miller is the only other guy who has access to it so we are hoping to preserve this pump data. If you would write to us or call Max and tell him what you would like to have, he will be able to explain to you what is around.

(McBride) On your second to last page, I see something that looks pretty familiar, which is your outlet axial velocity versus radius ratio. Down at the hub you are predicting essentially constant or slightly increasing, whereas your data of course shows a falloff. Is this because of curvature or just because of the error in loss, or have you looked into this?

(Serovy) I think it is because of an error in loss. I don't think it is curvature or anything like that. We don't really make any attempt to force it to go to zero at the wall. Then I guess I don't really know how to do that. I've got so many problems still existing in the center of the passage that sometimes we don't know how to get that.

(Lakshminarayana) One more question about the axial velocity; looking at the explanation in the theory, it looks to me as if it doesn't satisfy continuity.

(Serovy) It is possible that at one of these stations, the outlet distribution might not. The outlet distribution that is calculated is forced to be equal to the inlet flow rate. Remember the measurements that NASA made, the survey integrations don't always check the inlet flow rate, so if there is an error in flow rate it is in the experimental measurements. Because we force our computed values of flow rate to the entrance or like venturi flow rate. Am I saying that right? Do you understand?

(Smith) Let me call your attention to the fourth printed page. (Figure 6) There is something on it here, and I would like to make a point. Down in the lower left-hand corner there are some axial velocity distributions. I would like to stick my neck out and say that it looks to me as if the outer two-thirds of the annulus is casing boundary layer when the pump is throttled up close to stall. I don't see anything surprising in that with the kind of geometry that is in the pump (a relatively low aspect ratio, and a high radius ratio).

As some of you know, I believe we should relate casing boundary layer thickness to the staggered spacing between airfoils. Some other people like chord better for a characteristic length; and for the idea I am presenting right now, it doesn't matter. The point is, it isn't blade height. You shouldn't correlate wall boundary thicknesses with blade height.

I think you are always going to have trouble making universal correlations as long as per cent immersion from the casing wall or per cent annulus height is the principal independent variable you try to correlate with. The reason is that if you are trying to get a correlation that works for more than one passage aspect ratio, then values of constant per cent annulus height are going to be different depths into the wall boundary layer, the thickness of which is related to blade staggered spacing. So I would suggest that to come up with a more universal correlation, you should start measuring the distance from the wall in terms of some number of blade spacings rather than some percentage of passage height.

(Serovy) That is an excellent suggestion. This is kind of going along with what you did in your "Brown, Boveri paper". That, incidentally, what he just said is not contained in body but the ideas are in this "Brown, Boveri paper", which is in the book which Roy has. That is a good book to get and read, I think.

(Bullock) On Figure 9, what is Reference 7?

(Serovy) I didn't talk about that, but Reference 7 is, I think that goes back to that Erwin and somebody cascade testing technique. That is a NACA TR, about 1016, I think, way back. It is just to get an equivalent circulation. Incidentally, axial velocity ratio has been mentioned at this meeting. We have gone back and tried to mess around with all these theoretical methods proposed for correcting for axial velocity ratio, and we don't find many that hold up very well.

(Mikolajczak) I would like to comment on the velocity profile data. Leroy implied that the velocity defect represented a casing boundary layer. I consider that when a defect spreads over 2/3 of the passage, it is no longer a boundary layer and should not be correlated as a boundary layer.

(Smith) I showed some axial velocity profiles at the Brown, Boveri symposium for relatively high aspect ratio stages, and there is little question what is boundary layer there and what isn't. The profiles are fairly flat in the free-stream portion and there are two wall boundary layers where the profiles deviate in a boundary-layer-like fashion from those that you would calculate without any end loss considerations. I say that if we take one of these high aspect ratio stages that does clearly have a free stream and wall boundary layers, and we take the free stream part out by chopping the middle out of the blades and squeezing the hub and tip together, that we would ultimately end up with a profile that looks very much like Professor Serovy's. I therefore would call this outer part of the flow wall boundary layer, because it is in a region where there are strong three-dimensional effects, where the two-dimensional cascade data really shouldn't be expected to apply because secondary flow and tip clearance effects are large, and it is essentially end-wall dominated.

(Mikolajczak) The oversimplified correlation in terms of theta (θ) to the exponent "b" amazes me. The strong effects of velocity profile, axial velocity ratio, secondary flow and everything else are lumped together into the exponent. This oversimplified correlation will tend to push you into designing continuously for the same kind of velocity profile, which I presume you do not always want.

(Serovy) I agree with that. We put in this thing and this last loss correlation which you see merely as an effort to do the very best we could with limited time and to get some results before we quit working on pumps altogether. I think that my opinion about this whole program is that we have a tremendous main program for performance prediction. We run up and down these speed lines with the current correlations in a hurry. In other words, it takes to compute a complete performance map like a minute on the computer. It is the whole thing. It is really a fast,efficient program. The correlations that we put in, the subprograms that we put in, are still lousy. As you point out, we may be pushing ourselves in the wrong direction sometimes, and I think Roy's comment is excellent too. But I also observe that a lot of people are using things like this every day in design and analysis, and so we have to learn to get better correlations or to do something better in these subprogram areas in order to get ahead a little bit. Your comment is just right.

(Bullock) George, we have a message here. The conditions where your solution failed deserves more emphasis. My experience has shown that we get this result when the stream filament (or stream curvature) calculation techniques require the static pressure to be greater than the total pressure. Equilibrium demands this behavior. If you had used a stream function matrix, your details may have been fuzzy; but you probably would have determined the existence of an eddy. The flow is backwards, say near the tip, and forward elsewhere. When even the ideal flow fails to overcome the imposed static pressure gradient, a real flow will certainly fail. Notice, however, that this problem must be recognized in our development of turbomachinery requiring wide ranges of flow and high distortion tolerance. Developing the three-dimensional stream function matrix method should be pushed by all agencies that also commit money for compressor improvements.

(Serovy) Be sure you put that comment in the minutes. I would like to think about that some more. Mel and Jim know this. Where we get that loss of solution is surprisingly close to points in the experimental data where they just don't look too good in terms of their measurements in that region. Got any statement about that, Mel?

(Hartmann) I agree with what you have said. We are continually updating our design and analysis procedures. Jim has been working with stream filament methods to include the eddy. We still need to understand the necessary inputs to the programs to realistically fit the physical situations.

I would like to make a number of comments. Much of the credit for the consistency of that program should be given to Don Sandercock. Secondly, we have a similar program which will start soon for latter stages of compressors, with high hub to tip ratios. This will cover quite a range of flow coefficient, loading, and other parameters. Returning to the data that has been used here, I presume that if anyone wants a copy the data tape can be supplied.

(Serovy) Yes, our dying gasp in the pump program is to try and get them to let us help them, cooperate with them, in preparing this data in some sort of library form so that you can pull out a report and pick out the raw data and reduce it for your own requirements at some time in the future. There is a lot of good information here, and I surely would hate to see it get away. As I say, we have it on tape now. The data really belongs to them, to Mel and Don Sandercock and all these guys who have worked on this program. But we have it on tape, raw data, and we have it fixed up so that we can pull out anything we want and reduce it any way we want it. It is sort of a data file such as we have been talking about for future purposes in the compressor business. This is something that is really needed.

(Fagan) George, looking at the table of loss correlation equations, the (θ/c) is the momentum thickness at the trailing edge normalized by the chord?

(Serovy) That is right. Yes.

(Fagan) Let me hypothesize a very simple case where we have a nearly constant hub-tip ratio in a stator instead of a rotor. You wanted to know theta at the trailing edge. I would propose that you have to do a lot more calculation than you are doing now, but you go through a boundary layer calculation and calculate it. It is the kind of gap, between my boundary layer friends and the design friends here, but it seems that there are some places where we can apply the boundary layer theories. They have done something about longitudinal curvature, but it is enough work. You kind of always constrain these things that have to do only with the inlet and the exit flow conditions; and certainly from a computational system such as you have, that is the one that is right, but we know it is a function of many other variables.

How much more are you willing to put into your performance analysis system to get a better theta over c (θ/c)? Maybe you are limiting it by saying all I will look at are the inlet and exit flow conditions here or maybe one max condition.

(Serovy) How much more, I'm planning on putting in all the rest of my life and all the lives of the graduate students I can get.

(Lakshminarayana) The problem is the designer always wants a nice neat expression which they can put it in; you cannot always go to the boundary layer calculation.

(Fagan) Why not? Just go ahead and calculate theta at the trailing edge. Again, I hypothesize a very simple case, only the stators and only a constant hub-tip ratio so we don't have three-dimensional or a lot of three-dimensional effects.

(Serovy) We have not tried this. I have nothing against this.

(Fagan) I think it is trying to reduce the gap between the designers and the boundary layer people. It seems there is a place where they have it; they are kind of holding it out on the platter and saying, designers, please look. I am not sure why it has not been grasped at already.

(Vavra) What can you do then in the rotor?

(Fagan) There are a lot of three-dimensional effects in the rotor. We can use the old correlations in the rotor,and do this in the stator and hope we did the stators better. I think we have a philosophical problem here that where we do have better technologies, for some reason, they aren't incorporated because they aren't all inclusive. Because you cannot use it in both the rotor and the stator,we don't use it. Maybe I just don't understand it. In certain cases we do use it. It appears that we don't incorporate all the capabilities for a boundary layer calculation that we have today.

(Mikolajczak) If you take George's data and do a boundary layer calculation, you will probably predict the lowest loss point only. Elsewhere you will probably fail because the three-dimensional wall effects dominate.

(Fagan) I agree on that. But I tried to hypothesize another case where we really probably could calculate the losses as a function of other things. It is not this set of data. It is probably the least difficult part of the problem you have, maybe that is the answer to the question.

(Serovy) I was very glad to hear you say the other day that you have been able to take some boundary layer calculation system and get it to predict some sort of loss related to cascade losses.

(Fagan) For subsonic, but again it is not the one you really want. But for the subsonic situation it is rather good. It seems to be consistent; we haven't checked it out on a lot of data yet though. Consistently from the boundary layers in a wake calculation, which is only necessary because you made the measurements downstream a ways, agrees with the omega bars fairly well.

(Serovy) That's another feeler. As you say, another feeler going out in the right direction. We have to do it sometime. I don't think it is possible to do it for these cases now. At least I don't think so, but I agree with the thinking that says let's go ahead and try it.

(Olson) It is easy to make the boundary layer calculation; but in order to do it, you have to have the pressure distribution. It seems to me that is the hardest thing to get.

(Fagan) That is the reason we said subsonic.

(Papailiou) As far as the two-dimensional loss calculation is concerned, one can do it. However, when one calculates flows in cascades one has to take into account the convergence introduced by the corner vortices; and there one can't do anything. In fact, secondary losses can't be calculated even in cascades. In my experience there are methods that can reproduce velocity distributions on two-dimensional cascades even with axial velocity ratios that differ from unity. Losses can be calculated with accuracy for the two-dimensional case and for the convergence and the curvature case. But one is stuck when one tries to calculate either the secondary loss or the convergence effects that are introduced by the corner vortices. As far as the deviation angle is concerned, in my experience it is mainly an inviscid effect. Calculations in two-dimensional cascades showed that the deviation angle could be reproduced through inviscid calculations by adding one or two degrees to account for the viscous effects. That is, the boundary layer, the viscous effect on the deviation was small so that it may be that even for three-dimensional calculations, by doing an inviscid calculation one can reproduce the deviation angle. I think that correlations ought to be improved in this respect by introducing corrections according to the design pattern one uses (for example, free vortex, forced vortex, etc.).

(Serovy) That is a big assignment; that is a hard job.

(Herring) It seems to me that you are being a little bit hard on the use of boundary layer methods to calculate loss distributions, because the first thing you say is that you can't get the end-wall effects and you can't get three-dimensionality. After all it is a two-dimensional method, and you are doing cascade experiments presumably to get two-dimensional results. It seems roughly comparable and much cheaper to make a numerical experiment than it is to do another cascade test.

(Mikolajczak) It would be fair to say that the two-dimensional loss is not the major problem; you can get around it by either calculation or cascade tests. Some companies have acquired this information over the years.

(Smith) The criticism is that that isn't where the problem lies any more, except possibly in the turbine cooling area. I think in the compressor area the two-dimensional solution is well enough understood that we don't have those kinds of problems any more. Where we have to make progress now is on the three-dimensional problems, which really are setting the frontiers of our knowledge.

(Hartmann) Doesn't everybody use a boundary layer calculation when he gets into something that is strictly two-dimensional, or when he is looking at something when he has no data and is using the correlation system? Tandem blades, everyone who works with us on this has used the boundary layer calculation for comparative purposes, so I think we are leaving the wrong impression. We do use it, but the designer is always going to go back to the correlations when he has data that fits his case and when he gets a strong three-dimensional case. We are using it, it is just that we have not come up with the correlation system or relation system to compare calculations to calculations.

(Huffman) I think most of the work that is being done that I'm aware of on boundary layer techniques is to extend them to three-dimensional flows. The final step is to extend them to cases where there are significant cross flows. I don't think this is very far off. To have methods that will do this is just a matter of months now. The other thing is that I think in general in the theory you can certainly move faster than you can experimentally, mainly because you don't have the problem with hardware, etc. Granted, the results you get prior to experiment may not be very good, but you can build a framework in general; you can build some kind of framework that you work with then.

(Smith) I would like to suggest that, in the compressor area, some challenging problems now are associated with the end-wall boundary layers and the effects on them of blade boundary layers, clearances, and shroud leakages. I would like to propose that some research work be sponsored to get some good measurements of the type that the boundary layer people need to base their analyses on, in the compressor environment. I mean in the actual stage environment, and furthermore I mean in the multistage environment. I would suggest that research compressors be dedicated to this activity and that some detailed measurements be made; and I recommend this as one item that ought to go on the list of things that this group recommends should be sponsored. The work that was done at Cal Tech back in the late '40's by Professor Rannie and his students was work of the type I am referring to now. In fact, I was surprised to see Professor Rannie's compressor over in your laboratory. I never knew what happened to it, but there it is, and a machine like that is easy to re-blade. I would suggest that three stages are a minimum; I would like to see four, five or six. It is a relatively big machine so it shouldn't be too difficult to get good flow measurements from it. There is no reason why the casing can't be made transparent so that you can look in and see the kind of flow that we are really dealing with. As most of you know, we have one that is five feet in diameter, and I am sure others have more or less similar machines. I think this is the kind of facility that should be used for three-dimensional boundary layer work. When you get looking in there, you see that the flow is quite three-dimensional, quite unsteady (at least as you approach stall); and mastering it is just a tremendous task.

(Vavra) I am very sorry that Professor Traupel, whom you have met at Brown, Boveri, couldn't make it here because he had some conflict with his teaching schedule. He is a purist, as you know, a tremendous theoretician; and he would have liked nothing better than to try to solve these problems on paper. He also came to the conclusion that it is not possible in the lifetime of a man to do this, so he has done experiments in turbines, multistage, large-size turbines. I think the work which he is doing goes along the same line. You see this everywhere that we have to live in the real world, as you said. We have to know what goes on, I think, before we can do any reasonable modeling which then can be used for boundary layer calculations.

(Fuhs) Dr. Smith suggested that in the research compressors he wants to use flow visualization for the secondary flows, but I think that the density changes and the pressures are such that this may be very difficult. The scale of the things are small and then the fractional changes in density, so that probably one will be forced into something like hot wires or some other technique.

(Smith) Of course, my comments were aimed at low Mach number tests where this kind of visualization is possible. As we get to higher Mach numbers this becomes much more difficult.

(Fuhs) That's the point that I make, at the low Mach number we have a small scale phenomenon, that is the secondary flows are small scale; you have low Mach number and when you put those two together, you don't have anything to see that you can see by flow visualization.

(Smith) I don't associate the Mach number with the scale of the phenomenon. At higher Mach numbers the details are certainly different, and there are other mechanisms. But I think end-wall boundary layer characteristics are substantially the same at low Mach numbers and at the Mach numbers that we are at now in our engines. Certainly in the rear stages of multistage machines, the Mach numbers are all subsonic; and I think that incompressible research is meaningful for studying effects which are essentially viscous, like end-wall boundary layers.

(Fuhs) If you go and look at shadowgraph, Schlieren, or interferometry in this research compressor, it won't do much good because of the fact that you don't have big density changes. If you don't inject helium or something like that you aren't going to see anything. The gradients are such that the scale and so forth; you don't see very much. I think that flow visualization in a low Mach number research compressor probably isn't as fruitful as other techniques.

(Bullock) I bet you would be positively amazed at what you would see in a low speed, big compressor if you blew a little cigar smoke in there. I think it would be very illuminating and something that would greatly benefit the people who have to work with the details of the flow structure, and I heartily endorse Roy's suggestion.

(McBride) You probably should not exclude the possibility of a water compressor for the flow visualization.

(Mikolajczak) I would like to suggest that we should not overlook cascades before passing on to compressor tests. We have learned a lot about what happens on the walls by starting with cascades. The basic flow on the wall, including cross flows, is not fully understood. Our correlations still have pieces missing. We can make significant progress by concentrating initially on cross flows. The cascades can provide us with some insight here which should be helpful in sorting out the complications of row interactions and unsteady effects of multistage environment. I agree that we also need good multistage information.

(Foley) There have been a lot of techniques developed to put velocity gradients into the uniform stream and tailor them to those gradients that you see in a compressor, at least to correspond to flow coming into the stators. You can then look at a flow as being a non-uniform total head flow being turned by a stage. We have gone to actually moving walls on cascades, as well as velocity gradients. You can do all of these things in an environment where it is much easier to probe and understand secondary flows.

(Smith) I don't recommend that approach. I think the best way to simulate the kind of turbulent structure that comes into blading is to make it the way it actually occurs in nature, by having other blading in front of it. I don't agree that it is difficult to do the kind of research I am suggesting, at least if you limit yourself to relatively low speeds. Making these measurements isn't all that difficult in actual running stages. It is my belief that you just can't simulate in a stationary frame of reference the kinds of flows that are generated by a rotor. You can get the time-average distortion and put screens or nails or something like that in to simulate the kind of total-pressure profile coming in, but it is a highly unsteady flow in the real world; and its unsteady structure is just different than you are going to obtain with devices such as screens.

(Vavra) Especially now after what Alex showed yesterday with these shedding wakes. Things like this you couldn't very well simulate.

(Foley) Flow visualization is extremely difficult in the real world, even in water flow. We can make rotating machinery with water flow for visualization. This is sufficiently turbulent (obviously, that is the nature of these flows), that it is extremely difficult to do flow visualization. So you are forced, if you want to get some feel for the secondary flow phenomena, to work in an environment that has reduced turbulence so that you can actually get flow visualization. One can, of course, use hot wires and such; and that works fine.

(Bullock) Flow visualization is just one of the several tools that have to be used. It helps one develop a model, or it helps one to strategically locate his instruments. A good analogy is one of the techniques used to investigate blade vibrations. A visualization technique - sand pattern tests - is used to locate the nodes. This, in turn, is a guide to locating strain gauges which tell us the magnitude of the stresses.

One must doubt that a comprehensive picture of the flow can be made by any technique. The picture must then be put together by bits and pieces. I personally advocate low-speed visualization or any other technique in order to assemble a tentative model, to at least initially identify what should be measured. Because we don't know what to study, we are flying blind.

(Olson) I want to go back to the boundary layer a bit. I said before that in order to use boundary layer techniques, we have to have the pressure distribution. Most of the interesting problems, particularly case boundary layer problems, that we are dealing with in compressors are strong interaction problems, that is, the pressure distribution is a result of the interaction between the boundary layer and the inviscid flow. So, in developing boundary layer techniques, I would just like to make the point that we should think a bit about how these boundary layer techniques are going to be used, recognizing that we are really dealing with strong interaction problems. I think that in some cases sufficient thought really hasn't been given to how these boundary layer procedures are going to be used once we have them. I think there is a lot of work that could be done in looking at how the strong interaction problems can be treated.

(Huffman) I think that's been done. At least it has been started by Spalding, because he has used essentially the same empirical input for both the boundary layer equations and the Navier-Stokes equations. So he has the capability of essentially handling strong interactions.

(Olson) That is one approach to take; unfortunately, it is one that requires considerably more computing time than I think the designer wants to use, and there are other ways. There are other ways to treat this strong interaction problem besides Spalding's.

(Herring) Would you care to comment on what some of those are?

(Olson) We are working on one of these. I don't want to comment on it right now, but I think very soon we should be publishing some of this work. We have done the computing of the annular diffuser case, cases up even to the point where the boundary layers merge, and have gotten very good results. This is not a procedure as lengthy as Spalding's Navier-Stokes solution. It is not a procedure in which we iterate between the inviscid flow and the boundary layer.

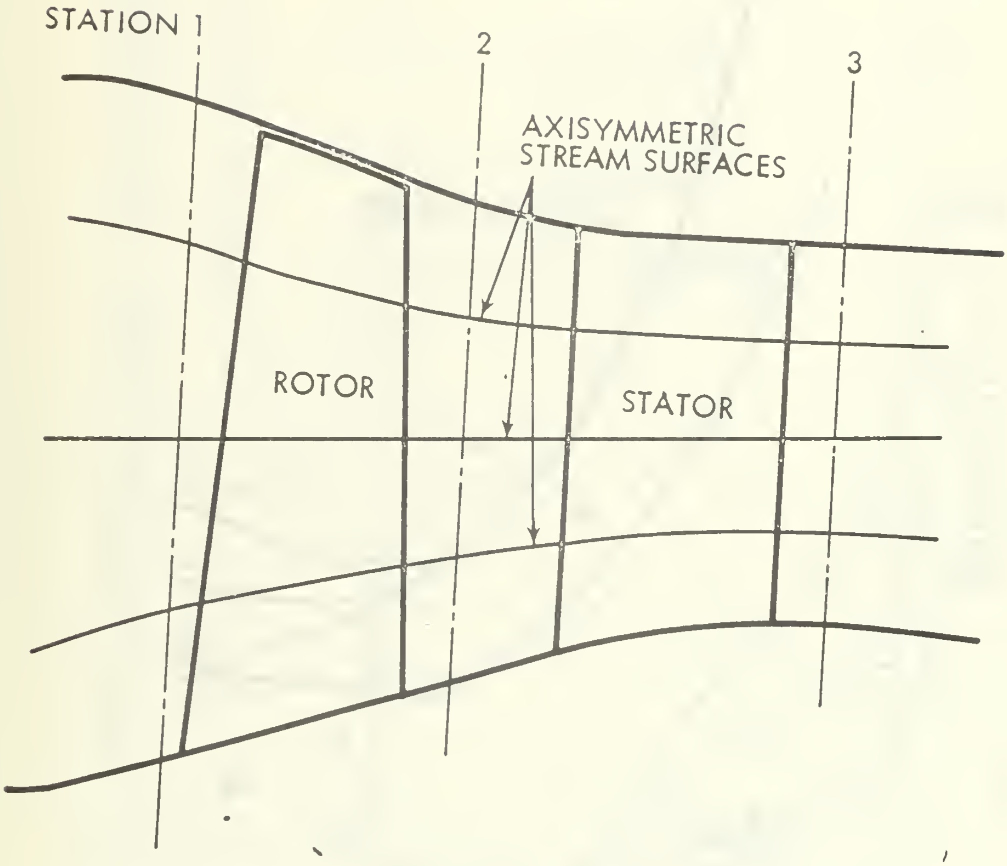


Figure 1. Axisymmetric, steady-flow model for axial-flow compressor design system.

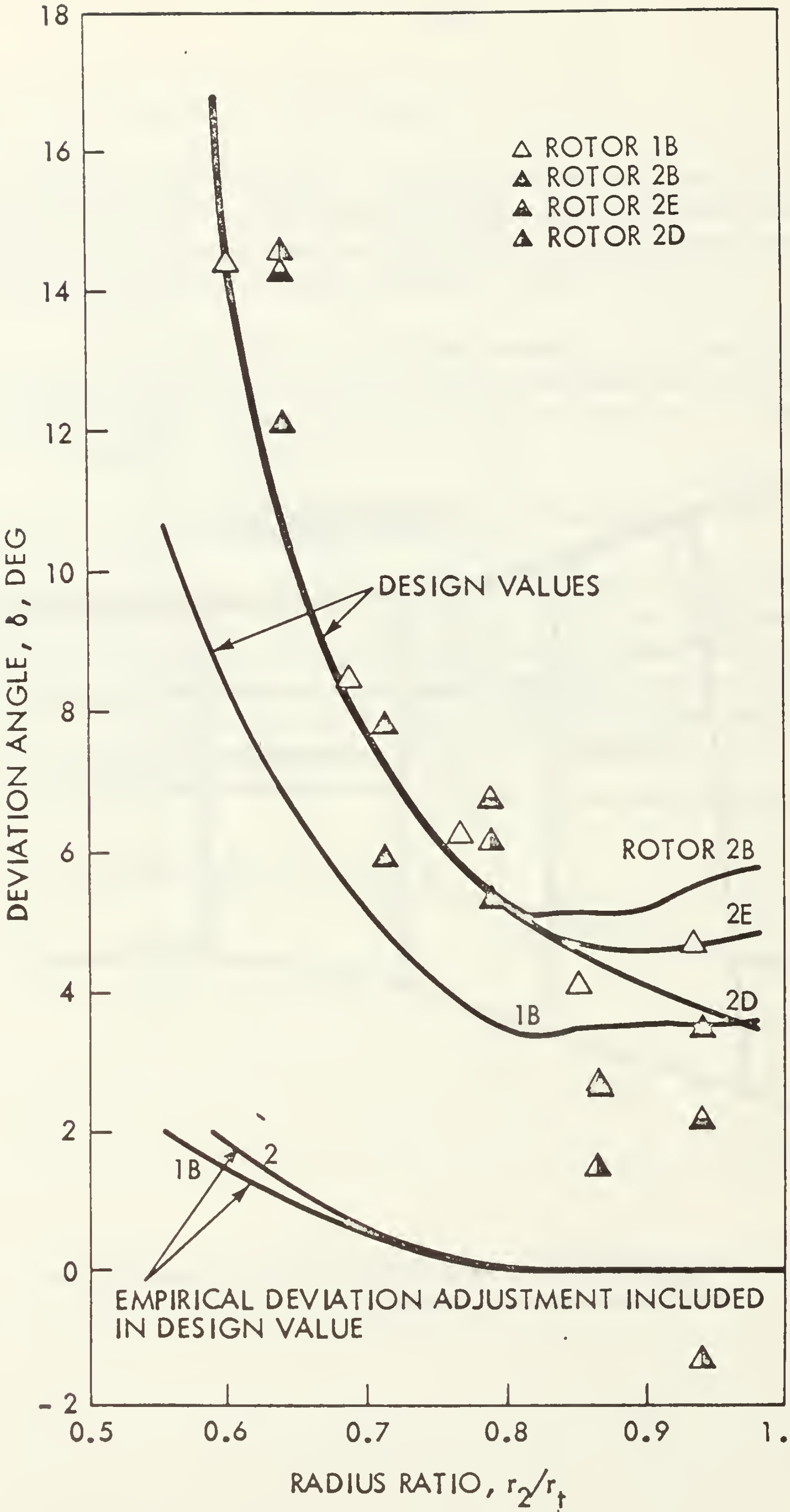


Figure 2. Comparison of design and measured values of deviation angle for four axial-flow compressor rotor rows (from NASA CR-1256).

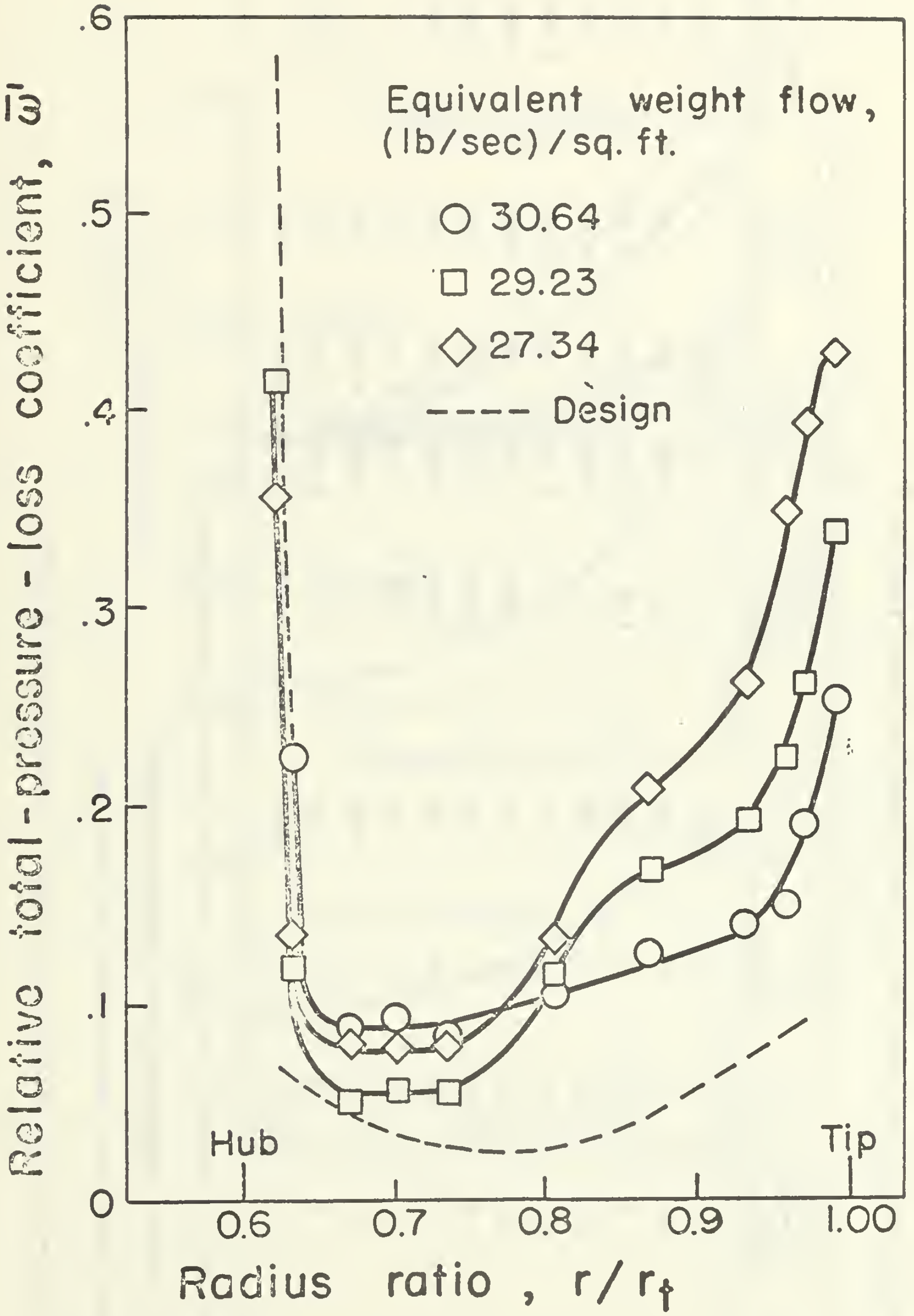


Figure 3. Comparison of design and measured values of rotor loss coefficient for a range of flow rates from an axial-flow compressor test program (from NACA RME56D27).

Table 1. NASA axial-flow pump rotor configuration descriptions.[a]

NASA configuration number	Tip diameter, inches	r_H/r_T	Number of blades	Blade section profile[b]	Blade chord length,[c] inches	Radial tip clearance,[d] inches	Design tip section D-factor	Design point flow coefficient
02*	9.0	0.4	16	DCA	1.5	0.013 – 0.020	0.23	0.293
07*	9.0	0.7	19	DCA	1.5	0.005 – 0.012	0.43	0.294
09	9.0	0.7	8	DCA	3.04	0.013 – 0.020	0.46	0.294
5*	9.0	0.8	19	DCA	1.5	0.015 – 0.017	0.66	0.466
6	9.0	0.8	19	DCA	1.5	0.025 – 0.027	0.66	0.466
8	5.0	0.8	19	DCA	0.834	0.007 – 0.009	0.66	0.466
9	5.0	0.8	19	DCA	0.834	0.015 – 0.017	0.66	0.466
10	5.0	0.8	19	DCA	0.834	0.022 – 0.024	0.66	0.466
13*	9.0	0.85	33	DCA	1.172	0.009 – 0.011	0.72	0.5
14*	9.0	0.9	19	DCA	1.5	0.009 – 0.011	0.63	0.7
15	9.0	0.8	19	DCA	1.5	0.009 – 0.010	0.55	0.466
16	9.0	0.85	33	CUBIC	1.172	0.009 – 0.011	0.72	0.5

[a] All rotors were tested without inlet guide vanes and downstream stator blades.

[b] DCA indicates a DOUBLE CIRCULAR ARC blade section profile.

[c] All blade chord lengths were uniform along the blade span.

[d] The range of circumferential variation of radial tip clearance is indicated.

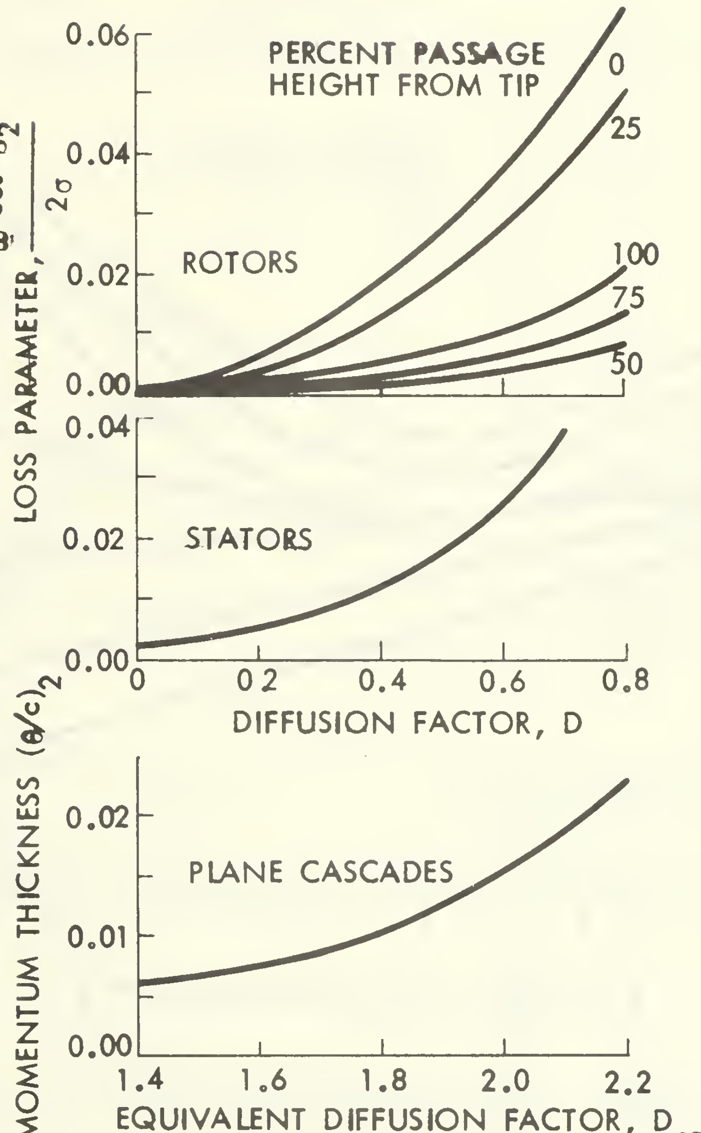


Figure 4. Loss correlation curves used in axial-flow pump design (from NASA CR-111574, 1970)

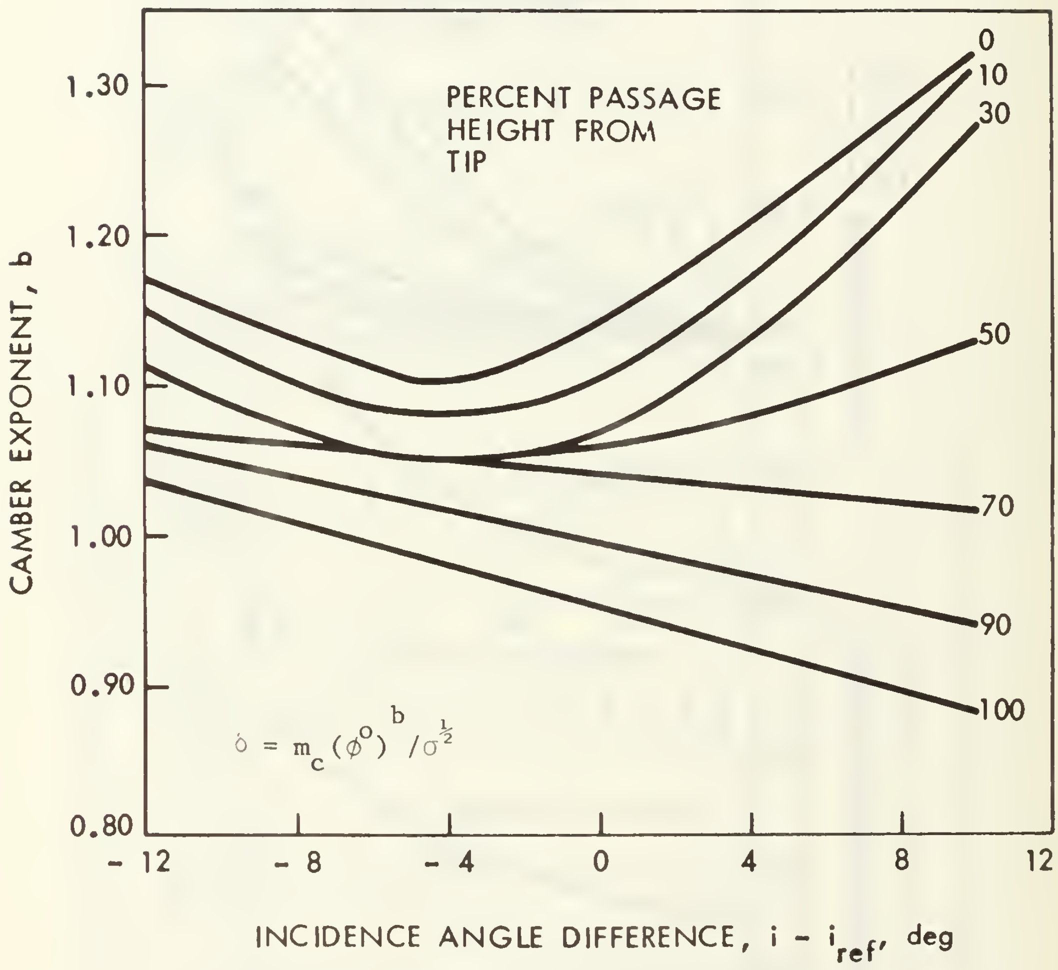


Figure 5. Deviation angle relationship derived from axial-flow pump data.

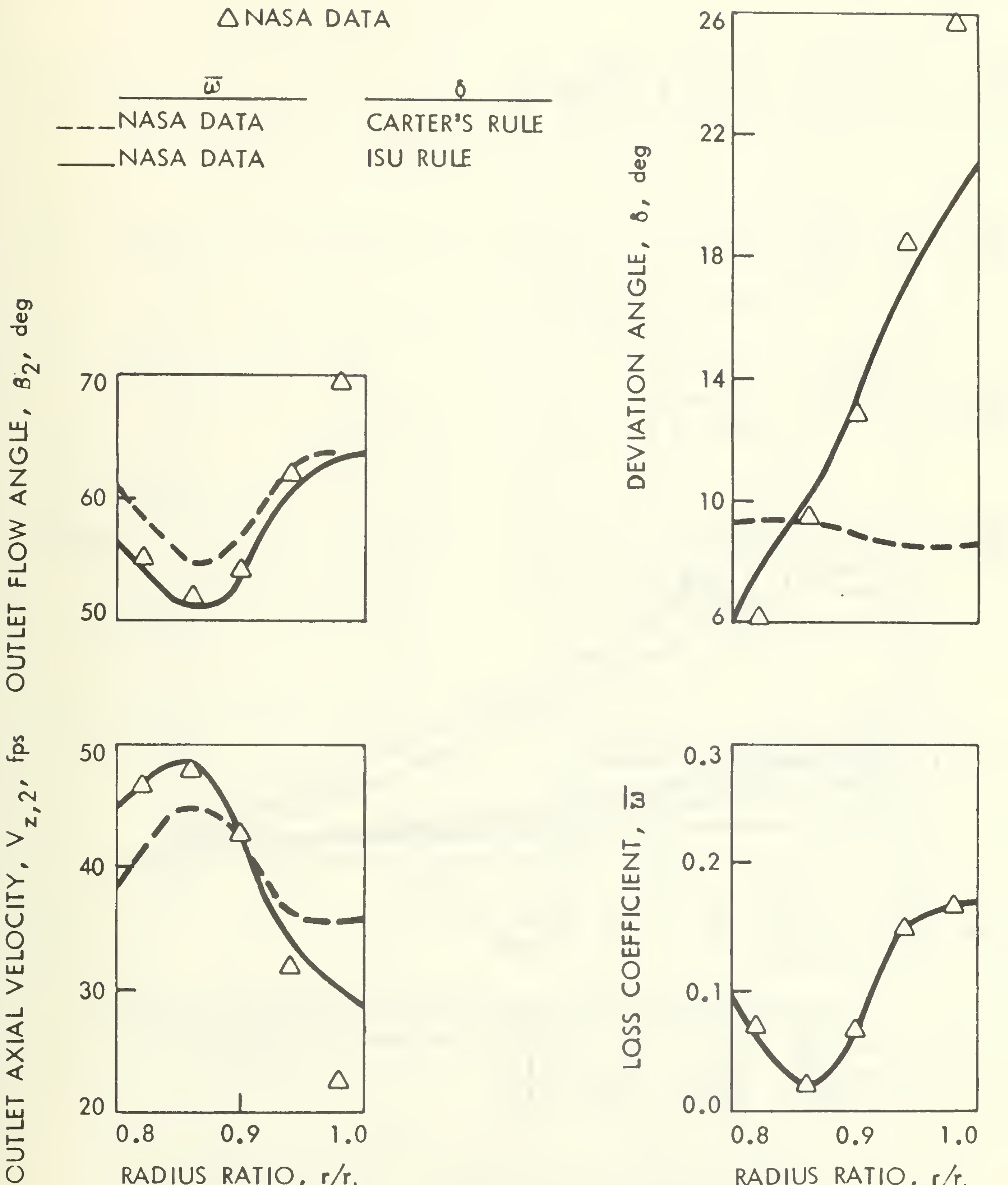


Figure 6. Rotor Performance Parameters for Configuration 15, Nonuniform Inlet Conditions 9 in., 19 Bladed, 0.8 Hub-Tip Radius Ratio Rotor, N = 3010 rpm, Q = 2853 gpm (ϕ = 0.338).

$$\left(\frac{\theta}{c}\right)_a = \frac{\bar{\omega}\cos\beta_2'}{2\sigma}$$

$$\left(\frac{\theta}{c}\right)_b = \frac{\bar{\omega}\cos^3\beta_2'}{2\sigma\cos^2\beta_1'}\left(\frac{V_{z,1}}{V_{z,2}}\right)^3\left(\frac{3\mathcal{H}-1}{2\mathcal{H}}\right)$$

$$\left(\frac{\theta}{c}\right)_c = \frac{\bar{\omega}}{2\sigma}\left(\frac{V_1'}{V_2'}\right)^2\cos\beta'_2$$

$$\left(\frac{\theta}{c}\right)_d = \frac{\bar{\omega}\cos\beta_2'}{2\sigma}\left(\frac{\cos\beta_2'}{\cos\beta_1'}\right)^2$$

$$\left(\frac{\theta}{c}\right)_e = \frac{\bar{\omega}\cos\beta_2'}{\left(\frac{P_1'-p_2}{P_1'-p_1}\right)\left(\frac{\sigma\,4\mathcal{H}}{3\mathcal{H}-1}\right)+\bar{\omega}\sigma\mathcal{H}}$$

$$D = 1 - \frac{V_2'}{V_1'} + \frac{\left|r_1V_{\theta,1} - r_2V_{\theta,2}'\right|}{\sigma V_1'(r_1+r_2)}$$

$$DEQ = \frac{V_{z,1}}{V_{z,2}}\frac{\cos\beta_2'}{\cos\beta_1'}\left\{C_1 + C_2\left|i - i^*_{2-D}\right|^{C_3}\right.$$

$$\left. + C_4\frac{\cos^2\beta_1'}{\sigma_1V_{z,1}}\left(V_{1,\theta}' - V_{2,\theta}'\frac{r_2}{r_1}\right)\right\}$$

Table 2. Loss Correlation parameter definitions.

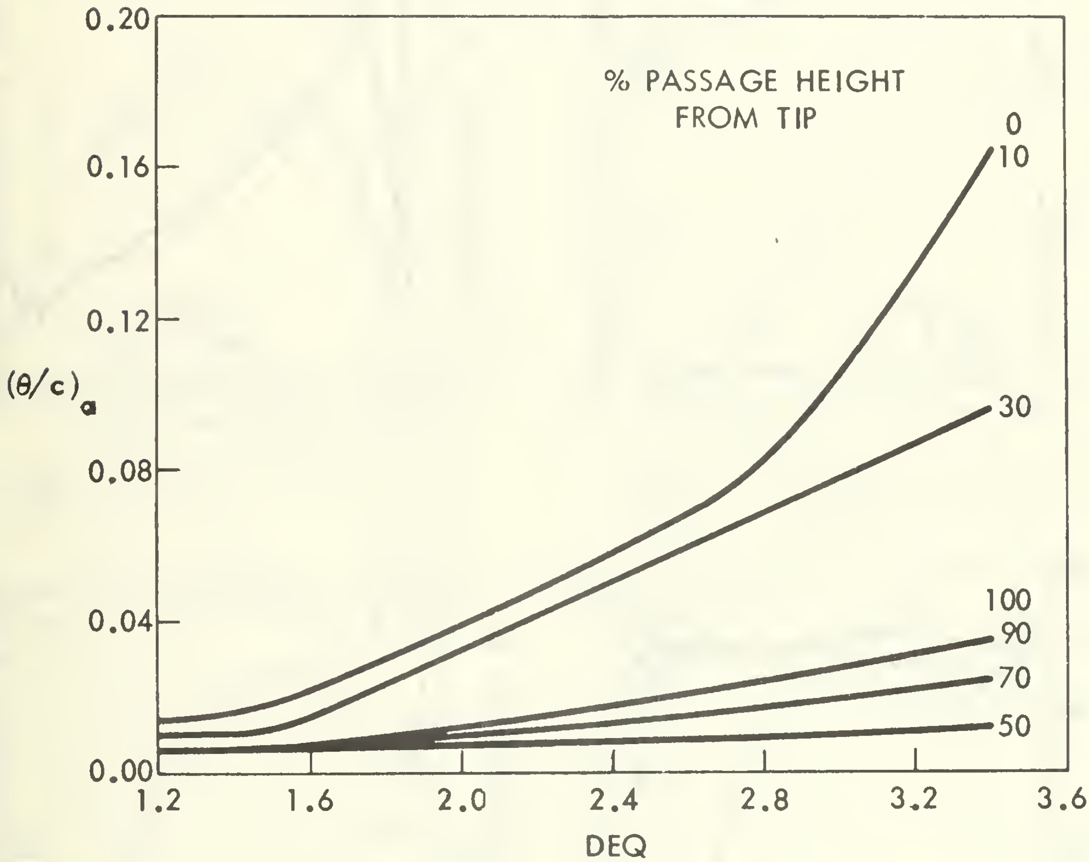


Figure 7. New Loss Correlation Based on Experimental Data for Pump Configurations 02, 07, 09, 5, 6, 13A, 14A.

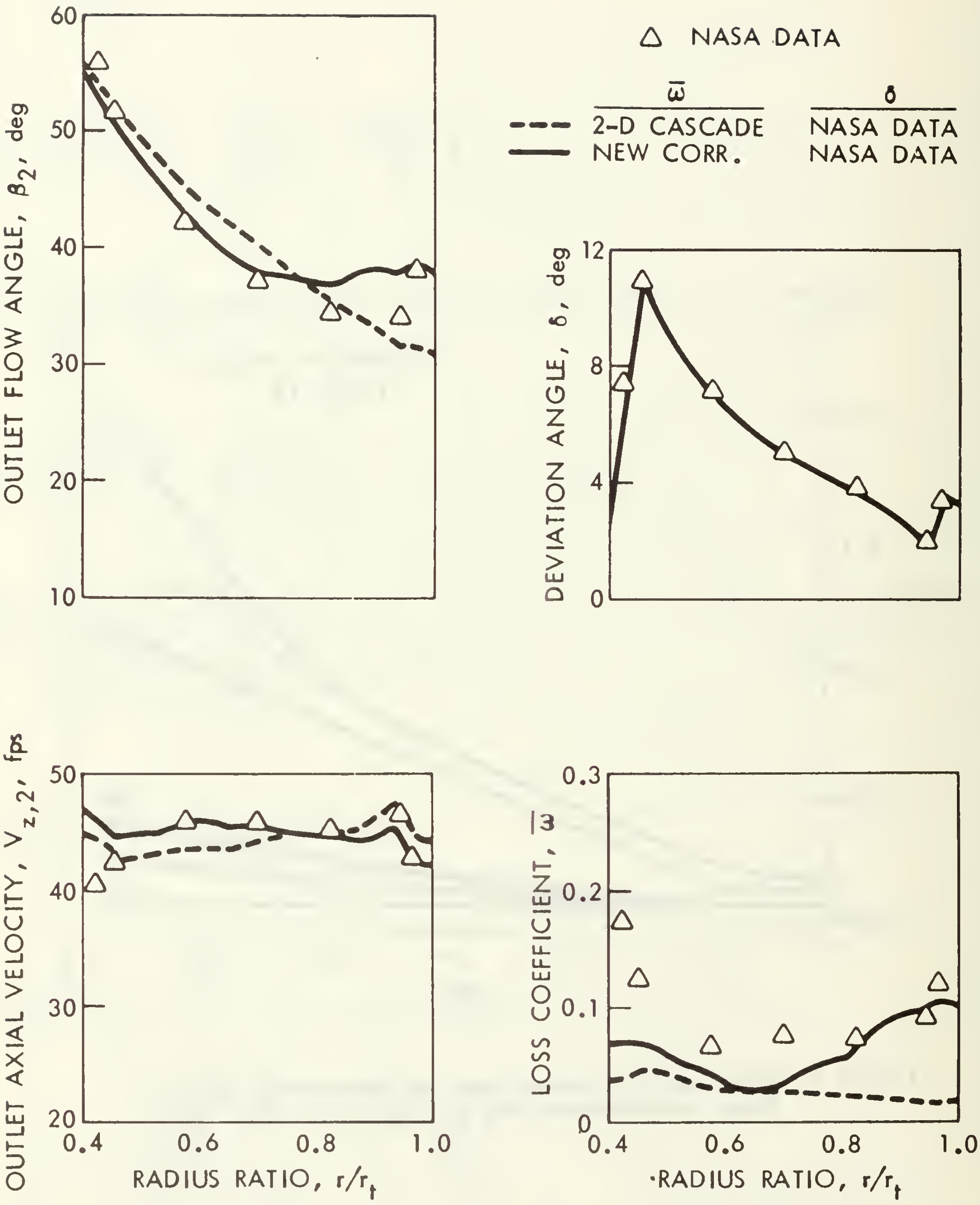


Figure 8. Rotor Performance Parameters for Configuration 01, Nonuniform Inlet Conditions 9 in., 19 Bladed, 0.4 Hub-Tip Radius Ratio, N = 3910 rpm, Q = 7446 gpm (φ = 0.291).

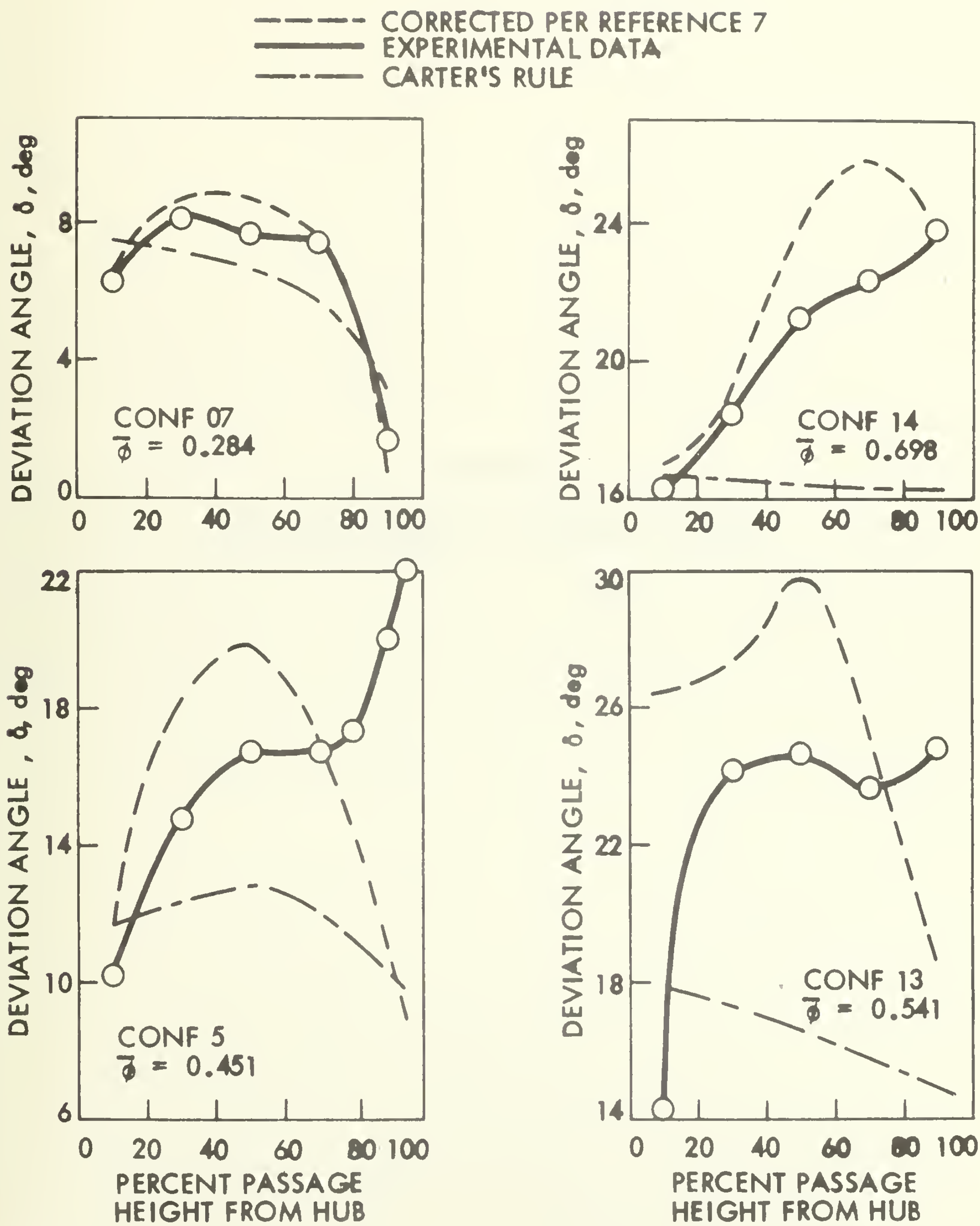


Figure 9. Results of using the axial velocity correction of NACA TR 1016 at design flow coefficient.

FLUTTER

Discussion Leader: Professor F. Sisto

Introductory Remarks on Flutter
by
F. Sisto

I think the job that Professor Vavra just handed me in the program as sort of a last minute change is very similar to the position of the field that I want to talk about, namely aeroelasticity and flutter. In the overall turbomachinery design problem we were sort of brought in at the last minute, perhaps after the last minute. It is a sort of pessimistic field to begin with. We've got to put constraints on the aerodynamicist; stresses are very real things that limit our aspirations in terms of performance, speed, and efficiency. Flutter, the aeroelastic business, has a background in the airplane-wing problem which goes back to the early 1900's. It was of some advantage for the turbomachinery designer to have this background because it gave him an initial framework in which to think about flutter and aeroelasticity. It gave him the concept that perhaps he could predict the unsteady loading in terms of aerodynamic coefficients, which is a very real concept that helps to cement his thinking and organize his research.

There is a whole structural side of the problem that I probably won't get into here because this is a workshop on flow in turbomachinery, but certainly aeroelasticity is an intimate combination of both aerodynamics and structures. And on the structural side, the airplane-wing experience gave the framework to think of the vibrations in terms of vibration theory, the concept of modes, the principal bending or principal torsion of the wing. So we can think about torsion or bending of axial-flow blades as little wings. And the idea of a reduced frequency was one that, again, was initiated by that long background of wing problems. The reduced frequency is a dimensionless group that seems to help with all sorts of correlations and to pop out of any sort of analytical or theoretical development.

But there have been very serious drawbacks in having this field grounded in the airplane. The axial-flow machine is not an airplane, and it's typical blade is not an airplane wing. Perhaps its most significant feature is that it consists of a multiplicity of these little airfoils sticking out in some sort of annular array. When we get our thinking compartmentalized because of our airplane-wing background, we miss completely many important phenomena that have plagued us and which require new thinking to stimulate innovations. We missed rotating stall as a possibility until its destructive effects hit us in the face. We missed the concept that you have a stall flutter. When I say stall flutter, perhaps I should define that, but I will assume that this group has a pretty good idea what is involved there. The flow is stalled, but in a time-dependent way, so as to drive the vibrations. We missed completely the possibility of the bending stall flutter problem. The torsional motion was hinted to us by some earlier experience that resulted in tearing wings off planes; speed pull-ups which resulted in a torsional stall flutter. But the idea that stall could produce a bending response was a big surprise and it created quite some panic in the early '50's. The fact that we have a grid or cascade and multistaging introduced new forms of acoustic resonances where we have sound waves, shocks, bouncing back and forth between solid surfaces and with certain timing and geometric boundary conditions being satisfied you get a resonance situation. Well, unless you had analyzed a triplane or multiplane from that point of view, no one had ever thought of any phenomenon of that sort. Consider the whole concept of forced resonance; this is not flutter, but rather driving an airfoil into a large amplitude vibration by timing input in some way related to a natural frequency of the structure. We were, I think, led astray here also if we related our thinking back to airplane wings because the only important resonance there was due to the propeller passing by the wing. There the problem was relatively straightforward. Here in the turbomachine we have first of all two kinds of aerodynamic loads which are involved. This, I think, is something that has not been appreciated and which most people ignore and thereby oversimplify the problem. We have a force category of loading which is the true forcing of the airfoil. Say the compressor runs once a revolution behind six struts, and so the lift and the moment that the airfoil experiences fluctuate at that same frequency. But now the blade starts to vibrate because it has a fluctuating load, and additional loads come onto the airfoil due to the vibrations. If the blades were perfectly rigid and wouldn't vibrate, then they would feel no additional load, but they now start to vibrate. Completely aside from the fact that the vibration is being forced by aerodynamics, it is also developing additional secondary aerodynamic reactions by vibrating. The phasing of all these effects has to be taken into account. If these secondary aerodynamic reactions are sufficient to drive the vibration, then the initial forcing stimulus need not be present and a true flutter condition obtains. One then has to make a decision about a particular situation: whether we are in flutter or we are out. The machines make this recognition quite forcefully; it cannot ignore the physics of the situation. Failures have been quite common and longstanding.

So the airplane-wing background has been helpful, and it has also been a hindrance. I think probably the whole field of aeroelasticity related to turbomachines has only matured within the last four or five years, ten years at the most, to the point where we are moving into new areas free from our straight-jacket thinking that was the result of that particular background.

Where are we today? Let's categorize our abilities first in subsonic flow and then in supersonic flow. Our present status is that subsonically the unstalled predictions are rather good I would say. We have very sophisticated programs based on low Mach number aerodynamics which allow us to recognize things like flow turning through the compressor stage and which allow us to recognize the three-dimensional nature of the structure, although we still persist in describing aerodynamics as a strip theory. Certainly we are having problems with nonsteady subsonic flow, particularly when you have low aspect ratio blades and a lot of the flow is corner flow or a lot of the flow is boundary layer flow.

We have been discussing here for the previous couple of days the difficulties of describing the flow when you have so much boundary influence. That must also be counted as a deficiency in an unsteady area. Even though I say that the state of the art is fairly good there, I am really talking about high aspect ratio blades that are not too heavily loaded and that are far from stall. So our prediction is good there; but, unfortunately, that is not where the major problems occur.

We have had a nice confirmation of the theory, however, in several ways, and there are situations where minor problems can occur. Coupling, for example, a very long fan blade with relatively weak disc such that the whole mode of the disc plus the blades, with a very clean attached flow far from stall, can produce a flutter condition. It hasn't proven to be a serious problem in the sense that we can recognize this and design for it. There are no, I would say, unexpected problems in that area. When the flow becomes stalled, I'm talking about heavily loaded sections, then the aerodynamics is again, I would say, in a satisfactory state from the point of view of predicting flutter. Non-stationary reactions to an assumed vibratory motion can be predicted well, but it is unfortunately based on a tremendous amount of empiricism. It is an area also of great corporate secrecy, I would say. Companies keep to themselves the great detail of the results of very long, extensive, expensive experimental programs that have delineated flutter boundaries almost entirely for those cases that have been tested. But the data "solve" the problem. I don't mean to berate that particular approach or pragmatic way to do it. It seems to have satisfied a need. However, when a new configuration comes along, one that hasn't been tested or one in which the flutter boundaries are not established empirically, we have a serious problem. I think a beginning has been made in that area. There is a little more theory, or at least a little more theoretical framework in which correlations can proceed, based on some analytical solutions that completely solve the problem of the fully separated cascade of oscillating airfoils. Use of the nonsteady aerodynamic coefficients in a stability analysis connected with some accurate estimate of the structural damping of the system provides what I understand are some very satisfactory correlations. This, of course, is quite gratifying because it seems to imply that some progress can be made in this way. One doesn't have to continue to rely for ever and ever on testing of essentially each new configuration you are interested in to find out whether or not it is unstable.

The forced vibrations in subsonic compressors is under an intense state of investigation at the present time in a number of laboratories and companies. The problem here I think is one of specifying the relationship between the response, the lift, and the moment in attached flow when the disturbance is specified. For instance, we had some discussion yesterday about the wake that is cut off between two blades in a subsequent stage behind the stage producing the wake and the shape of that wake, its relationship to the operating conditions of the blade producing the wake. These need definition, but certainly we need even more definition when trying to run an airfoil, or cascade of airfoils, through that wake and chop it up a bit. What are the responses in periodic lift and moment that are experienced by these airfoils? The problems there are rather intense because we have to make initially very gross assumptions about the flow. For example, we know it is a shear flow, and it is highly sheared certainly in the wake; we still have to use potential flow methods for the aerodynamics by and large. There are some few shear-flow aerodynamic solutions, but we are certainly not very skilled in treating lift and moment of an airfoil in sheared flow; and it is usually sheared along the span and not normal to the chord as is the case in running through a wake. So this intensive work that is going on is necessary and will, I think, ultimately allow us a quantitative ability to predict the amplitude of the vibration in the response mode that is produced by a given stimulus, which is not currently possible. I believe that the capability will be in hand very shortly.

In the supersonic area we have the availability today of analytically derived coefficients, unsteady coefficients; and to my knowledge, these are available for any regime below the hypersonic range. The methods that are available are general enough to treat all of these cases so long as the flow is unsteady and so long as the vibration frequency is high enough. One thing that I personally like, I think that the industries and the laboratories are beginning to get some experimental data in the areas of supersonic compressors, supersonic fan tips, etc.

One of the important outcomes, however, of this supersonic formulation of the unsteady aerodynamics, which as I said must be fed into a stability analysis along with structural information, is the apparent confluence of two fields of research; the one I have been describing, nonstationary aerodynamics, and on the other hand, noise production characteristics of turbomachines and their acoustic properties.

The problems involve phenomena such as the reflection and transmission characteristics of a cascade of airfoils impinged upon by a plane, acoustic wave. The solution of that problem gives you very similar kinds of overall domains of influence related to axial Mach number as one would derive coming in from the aerodynamic side when trying to predict the flutter coefficient. So I think we are seeing the growing together now of two separate fields of endeavor, and perhaps Dr. Foley will talk more about that later in the program.

The reason that the acoustics is important is not only because sound is produced, which can be annoying and produce some kinds of panel fatigue of lightly loaded skins, but also because very large structural loads, operating loads within the engine, are within the compressor where the noise is being generated. And certainly if the noise production spectrum is in a frequency range to which this structure can respond, we have a very serious situation. There are many indications that this is true.

Unexplained blade failures which seem to defy any sort of rational analysis on any other basis can probably be attributed to the acoustic type of resonances, which are very rich in number and possibilities and which, I believe, can be analyzed by pushing this particular field forward. This is one of the areas where I would conclude by recommending it as being a very fruitful area for future research and future investigation; bringing our acousticians together with our aerodynamicists and trying to formalize and increase our understanding of the detail of the behavior of blading systems as acoustic generators and acoustic filters.

One of the other things that I believe that is seriously needed is an investigation pointed at non-stationary aerodynamic coefficients for thick, highly cambered airfoils. I'm thinking now of turbines, turbine rotors and turbine stators. The nonsteady aerodynamics have not been analyzed by any of these techniques that I have been talking about with any accuracy because the techniques have all relied on thin, slightly cambered airfoils. What we need now is the solution of a problem that might be characterized by, say, a series of thick airfoils which have very high, steady Mach numbers, which one can presumably calculate reasonably well today, and which are now allowed to vibrate. I think we still have to confine ourselves to small amplitudes.

What is the perturbation on this high Mach number flow produced by a small periodic vibration of the boundary. I think the problem is a reasonable one. I'm not talking about one of these millennium type of problems; I'm talking about a look at this problem perhaps recasting over into some other plane, such as a velocity versus flow angle type of domain, and solving the problem with reasonably flexible techniques. That kind of technique has certainly been used to study similar problems, the small perturbation analysis.

I think that this high Mach number about blunt, thick, highly cambered airfoils has been fairly well put in hand in a steady flow, and I believe it is a very reasonable and desirable objective to now get the non-steady flow solution as a small perturbation on the mean flow. I don't know whether or not this has been done. I suspect it is being done in at least one laboratory or one industrial setting, but it isn't available.

Finally, there are, I suppose, other types of phenomena that I haven't treated in the supersonic field. This last problem is not necessarily supersonic, but it may be supersonic beyond the throat and it is certainly a high-Mach-number kind of flow. Another kind of supersonic phenomenon, that I believe can be shown to be troublesome and which needs investigation, is what I tend to call a bi-stable shock problem in that we have a series of airfoils that are vibrating relative to each other. Then it is clear that at some instant when the blades have a particular configuration with each other, there will be a certain stable shock configuration. But the blades at some other instant of time have vibrated to some other relative position, and the shock system may prefer to stand somewhere else in that distorted passage. So the shocks will be bouncing around in a sense somewhat similar to some aileron-buzz problems. And the two loadings on the airfoil represented by the two alternate shock configurations are such as to drive the vibrations. One could argue perhaps with the choice of words; and, furthermore, the physics may not be quite as clear-cut and simple as I have tried to make it in that simple abstraction that I just gave you.

I think that what I have tried to do is describe a historical progress in this field and tried to estimate very briefly where we are today. Dr. Mikolajczak from Pratt and Whitney is going to treat us to a film and perhaps some words along with the film.

(Fagan) May I ask a question? They have done a lot of work on unsteady aerodynamics but in fixed channels. Are there any comments about extending that to a situation where there are moving channels, for instance, slightly unsteady flow or moving the walls, and a numerical solution of unsteady fluid mechanics? Is any of that being done that you know of? Do you think that would be a fruitful way to go at the aeroelasticity problem?

(Sisto) I am not completely clear about your question. I know that in the field of fluid mechanics there is a current trend to solve very complicated steady-flow problems as initial-value problems. Is this what you had in mind?

(Fagan) Yes, that is exactly what I had in mind.

(Sisto) I think this technique is fruitful, yes. I think that is has been used by Giesing at Douglas to solve highly nonlinear interaction problems of airfoils. Of course, he has had to limit himself to small numbers of airfoils, like one airfoil passing another or a cascade of airfoils in small amplitude motion. He uses a sort of package routine and the problem is solved iteratively by taking small time steps. So it isn't often possible for him to come to a complete oscillatory-equilibrium kind of solution because he runs out of time steps. But he is able to track vortices, to track the boundary conditions on the mean surfaces that he has. But I don't feel that is the most important problem facing us; I think that is an isolated cleanup of the subsonic area. It improves our understanding and makes our coefficients much more precise. I was going to say that after Dr. Mikolajczak was through, I would certainly want to follow through on my duties as a discussion leader.

(Mikolajczak) Listening to George Serovy's very lucid expose of the design process, I noticed that he missed one major ingredient. Nowhere in his speech did aeroelasticity or mechanical integrity appear. When we carry out a design, even a preliminary design, we consider the mechanical aspects and iterate continuously between aerodynamics and stress requirements. We find that flowpath and airfoil geometries are often dictated by the durability requirements. For example, chord lengths may be set by flutter criteria. I hope this will reassure Dr. Sisto that flutter experts are called in at the early stages of the design and not only during panics.

Figure 2 shows a schematic representation of flutter boundaries on a compressor map. The subsonic flutter is quite well understood and can be predicted. At high speed we run into two possible flutter boundaries. Supersonic bending flutter has been occasionally observed, but has always occurred near the surge boundary and so has not been as much of a problem as the supersonic torsional flutter.

How can we move the supersonic torsional flutter boundary? We can increase the reduced frequency parameter (wb/W) by raising the torsional frequency (w). This usually requires a thickened blade and costs performance. We can increase the blade chord (b), but then we might exceed untwist stresses at the blade root. If the inlet velocity (W) is fixed, the only choice may be to stiffen the blade with a part-span shroud and live with an efficiency penalty. Other problems now arise. We can get second-mode coupled flutter of the entire blade-disc-shroud assembly. The coupled flutter is well in hand and has been discussed in a paper by Carta (Ref. 4). An alternate solution would be to use materials with higher strength-to-density ratio and so eliminate the need for the shrouds. This is the direction we are taking with composite materials.

Current trends in fan designs are illustrated in Figure 3. As you can see, in the last few years we have made some significant progress in technology. Large share of the credit is due to Mel Hartmann and his group at NASA, who have continued to explore high-speed and high-pressure-ratio fan stages. The latest of Hartmann's objectives is to build a 2200-ft/sec fan delivering a pressure ratio of about 2.6. You will notice that we do not pay a large penalty in efficiency for going up in pressure ratio per stage nor for going up in speed. Both high pressure ratio per stage and high speed are attractive for new engine designs. However, in the high speed regime we run out of our flutter experience and the validity of the existing flutter correlations.

The current, semi-empirical prediction systems are quite adequate for today's needs; but, by their nature, we are limited to our experience level. Since the designers tend to stay within their experience limits and since the conservative designs may penalize engine performance, we require a model to augment and extend these correlations.

We cannot rely on the wing-flutter experience since there is a strong interaction between blades in a compressor, as you will see in the movie. An extensive flutter program in transonic stages would be very costly. To formulate a supersonic flutter model we have, in addition to rig correlation work, a cascade program designed to look at flutter. We aim to get an insight into the interaction problem in a simple environment and then extend our predictions to rotors.

Figure 4 shows that in a steady flow we have reasonable agreement between cascade and rotor performance, as was discussed earlier in this meeting. The comparison shows the performance of a multiple-circular-arc blade tested at an inlet Mach number of 1.4. Cascade incidence (i) remained constant over a range of static pressure ratios. You notice that in the region where we have good correspondence between the cascade and rotor incidence, the cascade and rotor axial-velocity density ratio (Ω), the agreement between the cascade and rotor total pressure recovery (P_{T3}/P_{T1}) and flow turning (θ) is very good. In Ref. (1) we have shown similar agreement for three different blade sections. Encouraged by these results we felt that we could get a reasonable representation of the flutter problem in a cascade. Our results to date have shown good qualitative similarity to rotor behavior. For example, we observed the interblade phase angle to be between 70 and 80 degrees when flutter occurs, either in a cascade or in a rotor. Again we have seen that when we increase the pressure ratio at constant inlet Mach number (Figure 5) we can get out of supersonic torsional flutter in cascade and rotor. These results encourage us to pursue the program further.

Before I show you the flutter movie, I have to describe the tunnel in which it was taken (Figure 5). The flow is from right to left. Behind the blades we have a mirror on a steel plate, so that we have to use a double-pass Schlieren to visualize the flow. There is a suction plate on the lower wall to ensure that we do not get any reflected shocks going back into the cascade. There are scoops on either side of the cascade pack to remove the wall boundary layer to help us establish a periodic flow along the cascade. Whenever we take aerodynamic measurements, we insist that all leading-edge shock waves be parallel. The film will show that although we start with periodic inlet flow, as soon as we encounter flutter the periodicity condition is lost. For the flutter tests the attachment of the blades was different than is shown in Figure 5. The blades were cantilevered from the mirrored steel back plate. They were free to rotate in a set of bearings. A torsion bar (placed outside the tunnel) gave us the flexibility of making large changes in torsional frequency of all the blades simultaneously. There was also a provision to make small frequency adjustments of each blade separately. In the movie you will see a big blob at about the blade mid-chord where the attachment has come through the mirror back plate.

I have three short movie clips which I will show. The first one shows bending flutter. I have included it just to get you used to the idea of an interblade phase angle. This clip was filmed in a high-subsonic tunnel. The large amplitude blade motion, which appears to propagate as waves up and down the cascade, occurs in 1.35-inch chord steel blades.

(Lakshminarayana) What kind of inlet flow did you have?

(Mikolajczak) The inlet flow is at a Mach number of .75, and there is zero incidence on the blades. This is pure bending flutter and not stall flutter.

(Katsanis) Is this actual speed, not speeded up or slowed down?

(Mikolajczak) The film is slowed down. It was taken at about 3000 frames per second.

The next two clips show supersonic torsional flutter, and were taken in the supersonic tunnel I described earlier. The inlet Mach number is approximately 1.65. The blades are made of steel, have 3-inch chord and 4-inch span. Notice the large amplitude oscillation and the dramatically changing shock pattern in the blade passages. The reflected leading-edge shock swings about 30 percent of the chord during the oscillation. (Figure 6 illustrates this.) Please notice the phase difference from torsional frequency. Now, in the next clip, two of the blades were detuned by 5%. The flow has become much steadier, and the interaction between blades has been almost eliminated. The frequency of the oscillations you have just seen was approximately 450 cycles per second, the reduced frequency (wb/W) was approximately .25.

(Foley) You might note that by changing the operating conditions, you can excite bending flutter in the same set of blades. They are not normally operated there because they would fail.

(Mikolajczak) I tried to show you that supersonic flutter is quite a complex problem as evidenced by the wave pattern you have just seen. What are the benefits to the engine manufacturer from better understanding in this area?

Accurate knowledge of the flutter boundaries could reduce engine weight and improve engine performance because our designs would not be overconservative. And, of course, in situations where we cannot be overconservative we could reduce development time substantially if we could ensure a flutter-free first design.

DISCUSSION

(Sisto) I don't know how fertile the discussions are going to be. I might just comment on one point concerning the movie, which I hadn't seen before. This question of interblade phasing or the amount by which an airfoil leads or lags the vibration of its neighbor seems to be a fairly constant thing depending almost purely upon geometry. That is to say, you have a cascade of relatively flat blades that you see for the higher Mach numbers as in this cascade. The inter-blade phase angle tends to be about 180 degrees if you are such a poor designer that you don't use any stagger. And it works its way down through 90 to about 60 degrees for highly staggered airfoils. It seems to be independent of whether it is a bending vibration or a torsional vibration. Then if you think of that flat plate cascade as being a very poor turbine rather than a very poor compressor cascade, the phase angle goes from 180 back down to about 300 degrees. In other words, one associates the worse phasing condition, which produced the critical condition at the lowest Mach numbers let us say, very much with the stagger. This seems to be independent in many ways of the flow regime. So that these travelling waves that you observed are very characteristic of cascade flutter.

One could almost say that the vibration amplitudes could be constrained to be small or random in nature if you didn't have this phasing of the interblade motions. It is the one thing that allows an instability of the cascade to develop, where the single airfoil would be predicted to be quite safe.

(Vavra) I have grave doubts that you can model these things in the cascade. I just can't believe that. I agree with you that if you take unique incidence angles, you can find some correlations between cascade test data and rotor test data, always from this unique-incidence-angle concept. Now when I see these shock waves going around there, I have grave doubts if this will be the same in the real world. Those shock patterns must influence the vibration pattern. How close do you think you can really model that in the wind tunnel, predicting things from that as they occur in the real machine?

(Mikolajczak) When you get oscillations, blades in a rotor will also get oscillations of the leading-edge shocks.

(Vavra) But it has to be in a different way, the periodicity of the flow which is superimposed on this whole thing.

(Mikolajczak) I am not sure, you see, how different it is going to be.

(Vavra) Isn't that maybe just the point which should be checked before we allow too much work to go on in that direction? As I said, at the unique incidence I agree with you. We also know that compressors do not only work at a unique incidence angle. We extend that, and the angle is not unique any more.

(Foley) There is another way to look at that. That is that the flow in the cascade is a much easier one to model analytically. Thus we tend to compare the theoretical analysis with the cascade results, rather than with rotor results initially. I think insofar as developing some interaction model and then developing confidence that your model is responsive to those parameters which have a strong influence on flutter, the cascade tunnel is a very useful device. Also, since the tip regions of the rotor, of course, are going to be dominant in flutter, it is probably not as far off as you might think to use cascade results.

(Oates) I think this question is along the same line really. I am wondering about the validity of two-dimensional tests. It seems to me that terrific shock problems come about because of a relatively small area change that is changing the gas dynamics tremendously. Won't there be a great deal of three-dimensional relief in the real world?

(Sisto) I would think this is particularly true in the transonic range. I think the representative-section concept, to take some section of the airfoil as sort of an average amplitude and conduct sort of a two-dimensional analysis at a section that is not too close to the root and not too far from the tip, would help to give us predictions that are only of qualitative value but value nevertheless. One shouldn't refuse a result because it is only qualitative. I think Professor Vavra's question is undoubtedly true, that one cannot be quantitative on the basis of cascade tests. I also think the regime of flow, strangely enough, is one important question. I have little or no faith in any subsonic, rectilinear-cascade work as far as unsteady flows are concerned. It is virtually impossible to model the periodicity condition properly at the end walls of the cascade. You can see the phenomenon travel along an eleven-blade cascade, and it comes to the end wall, and it stops. Whereas, in the real machine it would come around. In completely supersonic flow we have the leading edge shock swallowed in those adjacent passages, and I feel a little bit more comfortable about saying there is some relationship between that cascade test and the equivalent rotor section for some particular data.

(Vavra) They are usually not swallowed in the designs which we are doing now, because the axial component is really subsonic. I think this puts quite a different slant on the picture.

(Sisto) Well, maybe a qualitative proof in the sense that it still swallows one of them but not the other one. You are confined in that situation to strong propagation effects only in one direction.

(Foley) This is certainly an area in which the literature is very sadly lacking, that is, in information on unsteady aerodynamics of blades in cascades. It is bad enough for isolated airfoils, but when you get to multiple blades, nothing exists. The analysis is particularly difficult because you shed wakes at the blades, of course, and they are unsteady. This unsteady vorticity interacts strongly with the other blades in your cascade. So you cannot do analytical work that considers just the blades, but you have to consider the blades and a system of wakes. If you forget the wakes, you are sadly deficient.

The people that are concerned with solving the steady flow through a two-dimensional blade row have an almost trivial problem compared to the unsteady-flow problem, even though we can't solve the steady one.

(Sisto) I think that I could stand here, and make a fairly long list of complications that we can't adequately handle by theory. I don't see any particular advantage in doing that. Certainly one of them is multistaging. None of the theory around and even none of the good correlations, I would say, would handle the problem of a buried stage, the stage that has one or two stages ahead of it and one or two stages behind it. Some beginnings have been done on that kind of problem for the subsonic and essentially incompressible case. I think if we want to talk about further problems and complications that haven't been discussed, we could simply list them for a long time without being able to say anything positive.

(Schwar) I am not a flutter specialist and I may ask a naive question, but I noticed in the film on supersonic flutter that the leading-edge shock and its reflection inside the cascade move almost exactly in phase with the change of the stagger angle of the cascade, that is, as if it would be a succession of quasi-stationary situations. I am just curious whether you would have expected that the behavior of the shock and its reflection in function of the changing stagger angle as it was, or if you would have expected a lag between what you see, a time lag between the movement of the shock and the movement of the blade? It seemed to me that it was a succession of quasi-stationary flow conditions. As soon as the stagger angle moved like that, then the shock moved at the same time in what you would expect would be a stationary condition.

(Mikolajczak) Just because the leading edge shock moves in this manner does not imply that the entire flow can be analyzed in a quasi-steady manner.

(Schwar) What is the frequency?

(Mikolajczak) The frequency is about 400 odd cycles per second.

(Schwar) So you would expect that the movement of the shock closely follows the movement of the blade?

(Foley) It is not like stall flutter where there is a significant lag.

(Sisto) I question whether you can visually detect that kind of lag. I would say that you can see the airfoil oscillating also periodically in time. But to couple the two statements, to say that the shock is about where you would expect it when the blades are in a particular instaneous configuration, is difficult.

(Schwar) The only thing I observe from the movie is that simultaneously with the movement of the shock, you could detect when the blade started to come back. That is, when the blade was at maximum incidence, you had the shock in a given position; and then when it started to move back, immediately the shock moved to its other position. That is the indication that I get that things were moving simultaneously in a quasi-stationary way.

(Mikolajczak) The leading-edge shock follows the blade motion quite closely because the shock must always turn the incident flow in such a way that at the leading edge of the blade the flow is parallel to the blade. Using a detailed quasi-steady analysis, it is possible to explain what happens at the leading edge of the blade; however, the quasi-steady analysis cannot be used to predict the pressure profile on the rest of the blade. Only an unsteady analysis is satisfactory there.

(Sisto) I think you could make a very general statement, which doesn't perhaps answer your question directly, but by and large it is true, that the time lags which are involved or the phase lags, I would say, tend to be quite small in all regimes. We are talking about phase angles that hardly exceed 30 or 40 degrees in many instances; however, there are exceptions. So it is just the fact that you do have a phase angle in many cases that you can explain a particular instability. A very small out-of-phase component can be troublesome. I think that is a fair enough statement. For example, what I am characterizing here is a typical result in classical incompressible flow, the Theodorsen function (see Fig. 1), which describes essentially the lag in some component of lift, let us say, in which this number here is 0.5 and this number is 1.0 and this can be, for all intents and purposes, thought of as a semi-circle to scale the problem properly. So the lift never lags much more than about this angle, let's say, from the motion which is producing that lift. These are complex numbers here; this is the real part and this is the imaginary part. So you see that this kind of phase angle, which is truly of aerodynamic origin, is always small, that is, with proper other conditions, and probably is what accounts for the instability. I don't think that in the movies we were looking at that, if this phase angle, for example, was 10, 15 or 20 degrees, you could have visually decided that the shocks started back at the instant that the slope started to change then to 5 or 10 degrees in time later.

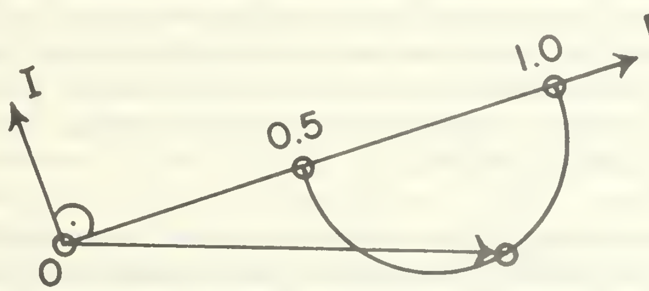


Figure I.

I tried to squeeze in some recommendations in my initial remarks. I think one of them that I would like to get some response from the assembled group here is the one I feel is in some ways closely related, and that is the coalescence here of the body of knowledge and research that has been done in noise, the acoustic generation characteristics of turbomachinery, and the very similar kind of formulation that seems to be growing up in the aeroelastic or unsteady aerodynamic field. I think there are areas of ignorance in both of these fields, they are both moving forward, and I think they would benefit very strongly by some generalized work that would include all possibilities. For example, for the noise people, people who are coming at this problem from the point of view of being able to predict the radiation characteristics from the fan stage -- are they satisfied with their capabilities? Do they see how they are going to improve their predictive capability particularly when a certain amount of the noise might be, perhaps, coming from vibration, not only wake chopping frequencies and other natural occurring frequencies, but also the fact that the blades are not stationary in a vibratory sense? We know certainly that going way back to the late '50's, the early '60's and right on through, there is hardly ever any trouble telling when a compressor is fluttering. A lot of that vibratory energy getting into the air is acoustic energy.

(Foley) I just want to make one comment along that line. When we were, in the recent past, establishing an acoustics group, we felt sufficiently strongly about the close ties between the two fields that we used the aeroelastic group, which already was in existence, as a nucleus in our acoustics group. Indeed, for much of the aerodynamics actually used to predict acoustics, one obviously wants to know the unsteady aerodynamics of the blading. That is, to predict acoustic effect, one obviously has to be able to predict the unsteady aerodynamics of the blades; and that is certainly true also in predicting aeroelastic effects. So there is an extremely close tie between both fields. To my knowledge, no one at the present time is really looking for very small amplitude vibrations of blades and trying to predict acoustic radiation therefrom. Although, if you give me the motion of the blade, I think one can fairly readily predict the radiation. More important is the aerodynamic response of blading to unsteadiness, as in the wakes and so on. The time-dependent loading which is needed for both flutter and for acoustics cannot be calculated at the present time. For even the simple problem of a cascade with axial perturbations, high frequency perturbations in velocity, I don't think we can do an adequate job.

We've got one predictive technique for variations in angle of incidence with constant dynamic pressure, and we have done some recent experiments to see if we can verify the analytical predictions. We find some moderate difference between the phase angle predicted by the theory and what we see experimentally for a blade oscillating in pitch. The differences in the magnitudes of the forces differ significantly. The information, I think, is relevant to both fields. But I do not think that at the present time much of the analytical work that is going on is pursuing different trends. The aeroelastic work is still linearized. In much of the important acoustic work, we are using non-linear aerodynamics, especially where there are shock waves.

(Sisto) My point is that I think we need to improve the aerodynamics in very much the way you have just described it. We have to presume to recognize their camber which you may possibly be doing in acoustics. But these things haven't been done yet on the aerodynamic side of the problem, at least in any systematized way that I know of. And by bringing the two fields together, it seems to me that they can influence each other. The rate of progress and the generalities of the results should be proof. There are many, many results, for example, in acoustics that have just no counterpart yet in aerodynamics. It should perhaps be there, such as interaction between stages.

The difference of the number of blades between a rotor and a stator, for example, is of no significance in the aerodynamics at the present moment; but in acoustics this is one of the important things that you look for to try and pick up a particular prime. Now that must mean that there is something deficient in the aerodynamics formulation; and I think that by trying to formulate the theory relevant to experiments that we conduct, we recognize that we are more or less treating the same kind of problem.

(Vavra) I think it is awfully hard for you to say from the top of your head what should be done, what sorts of programs should be undertaken. I feel that the need is here to get together, as you said, to treat the same sort of thing. I have had a very recent experience. You heard Professor Platzer talk about unsteady aerodynamics, and I want to corrupt him in the field of turbomachines. I could use his help. It was quite interesting how he looked at things entirely differently. When he saw shock patterns in front of a cascade, he said now is this true, the flow will not be disturbed? How far upstream will the flow be disturbed? And I said that as far as I know, the flow gets to the close vicinity of the rotor without apparent changes if you talk about design point; and then something happens in there. Then we came roughly to the discussion as to what is noise, and what are real flow deflections. How do these go together? I tried to impart my philosophy and experience to him, and we had a difficult time getting together. We were talking two different languages. Is this a matter of curriculum at schools? Is this a matter of undertaking programs where both specialists, you might say, get together and come to a formulation? This is terribly important, but I don't know how to do it.

(Sisto) I think the problem comes from premature formation of the model. One looks at the physics and right away we jump into a model as soon as we possibly can it seems, and we don't pause there very long. The model very often goes by the board, and we just look at the result. A typical example, I think, is that for years and years the supersonic aerodynamicists have been talking about sources. The aerodynamicist models his surfaces with supersonic sources. He gets the radiation pattern and tries to figure out where the influences have moved and what they carry with them, what effect they will encounter on other surfaces and so on. Only very recently did it occur to the aeroelastician that that source is on the airfoil and is radiating, and now the airfoil vibrates, and now you have a moving source. What does that mean -- a moving source? He didn't pause very long. He went on and formed his model, and he got his unsteady aerodynamic coefficient, but the steps between the modeling and the physics have always been done through very quickly. This is where, of course, the union of the two subjects would help, not in a mathematic sense but on a re-examination of the physics and what universal model will describe them both. I don't have the answer, but I think the proposal that I would make would be to model from the physics.

(Fagan) Might this be an argument for the unsteady flow with the vibrating channel approach to the problem? Even though, as you point out, it is probably not going to get to the answers of the real problems; but it has all the components of both those models in it. You see the whole flow field. You see the vibrating channels or surfaces. Model the surface as a boundary condition which is moving in time, oscillatory.

(Foley) You have to be careful. What you will hear as noise can arise because of some device that is oscillatory in nature, like the fluttering blade. It may also arise because you have a phenomenon which in the objects coordinate system is steady but to the observer is unsteady. If you have a wheel rotating and it has a shock pattern, to a stationary observer it will create noise. Thus, you can learn a lot about what you will interpret as acoustic phenomena from stationary sources where there are shock waves present; you don't have to flutter that cascade to learn a great deal about how the waves propagate out and create noise in the far field.

(Henderson) Dr. Sisto has made the suggestion here of combining the aeroelasticity approach to see what kind of noise generation comes from vibrating blades.

(Sisto) Well, to include that; I am saying the formulation would always take care of the steady acoustic radiation and the rotational motion.

(Foley) What I am saying is that there are many problems that you might interpret as acoustics that are steady in one frame of reference that won't require all this complex formulation.

(Sisto) I think if you want to get more sophisticated, you can also begin talking about larger amplitude disturbances where you get away from acoustics. This is certainly a fertile area, that is the one that does reach out to a millennium.

(Fagan) My thought is that in the kernel kind of approach, your source kind of approach, where you integrate over the surface and look for the result at one point somewhere else, you don't see the flow field. You lose a lot of information which is the tie-together between these; and if you took the unsteady flow problem, then you see the flow field which goes along with it. Now you can develop the flow field if you look for the answer at all the points, but we typically don't do that. But we really need to develop the flow field, a more complete picture of the flow field, to bring these things together.

(Sisto) One thing I wanted to emphasize that I passed over is of more than academic interest. It will open up the explanation of many types of instabilities that we see inside the engine.

(Fagan) We are specifically concerned with aerodynamics, but from the mechanical part of the blade-vibration problem is that well in hand?

(Sisto) No, I don't think the mechanical side is, but I eliminated that part of the problem from my discussion because of the nature of the workshop. There are many aspects of blade-vibration problems that have not been fully appreciated from the structural side that I don't propose to mention here. For example, one of the things that one has to look at very carefully is the phasing of the excitation along the radius. Very frequently the excitation of an airfoil is assumed to occur simultaneously in time at all radii, that is to say, it runs in and out of a disturbance at the root at the same time that it runs in and out at the tip. The fact is that in most situations one can show that this is not true, so we have essentially a traveling wave along a cantilever. That is, the load is not in phase all along the length of the beam, if you want to look at the structural problem. This is an anomalous behavior that is easy enough to explain once you formulate the problem that way. There are a number of more or less structural features of the problem that I don't feel are appropriate to go into now. That side of the problem is not completely satisfactory. I feel that the problems are pretty well divided.

FIGURE 2.

SCHEMATIC COMPRESSOR MAP SHOWING FLUTTER BOUNDARIES FOR THREE TYPES OF SIMPLE FIRST MODE FLUTTER

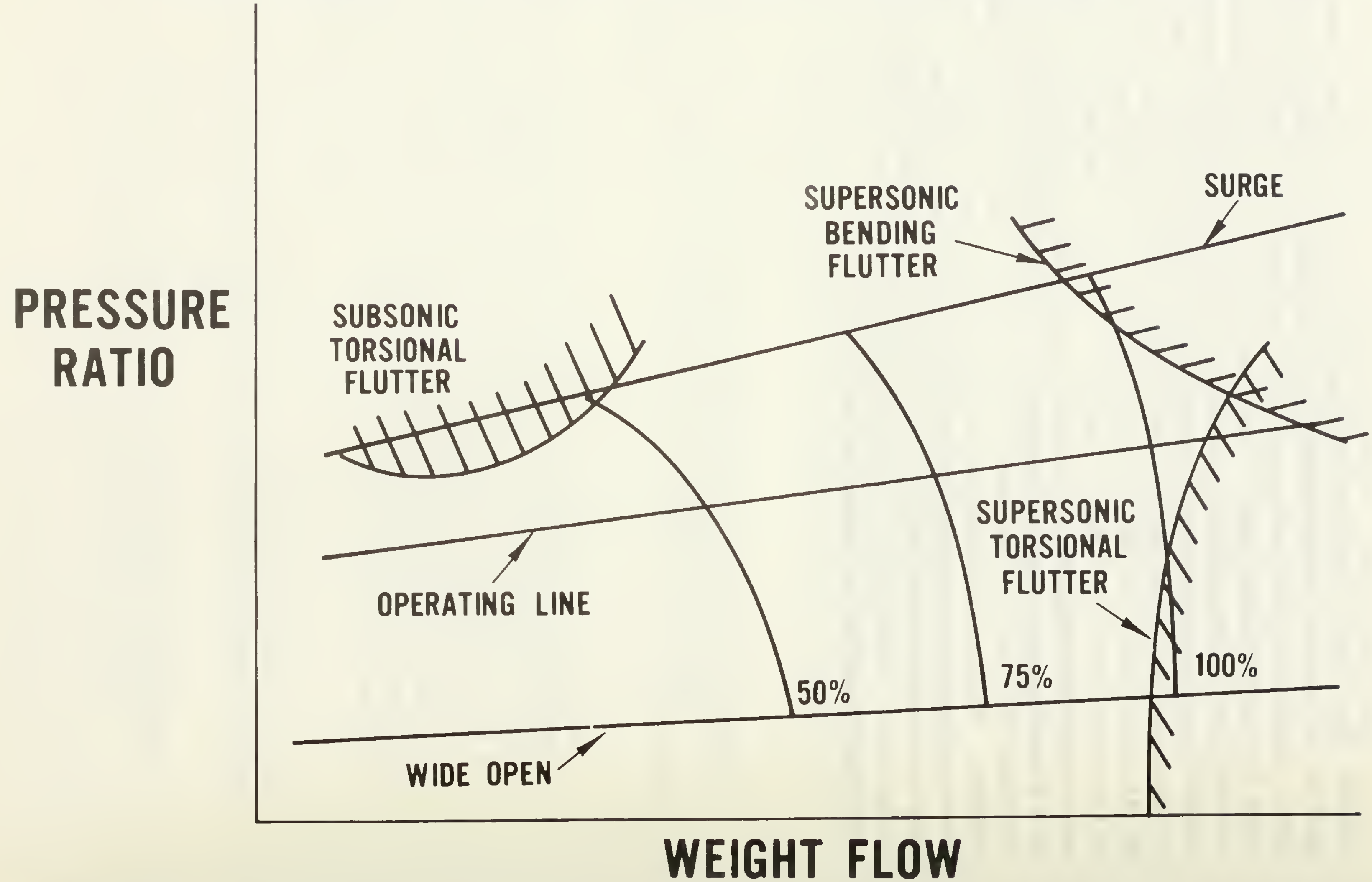
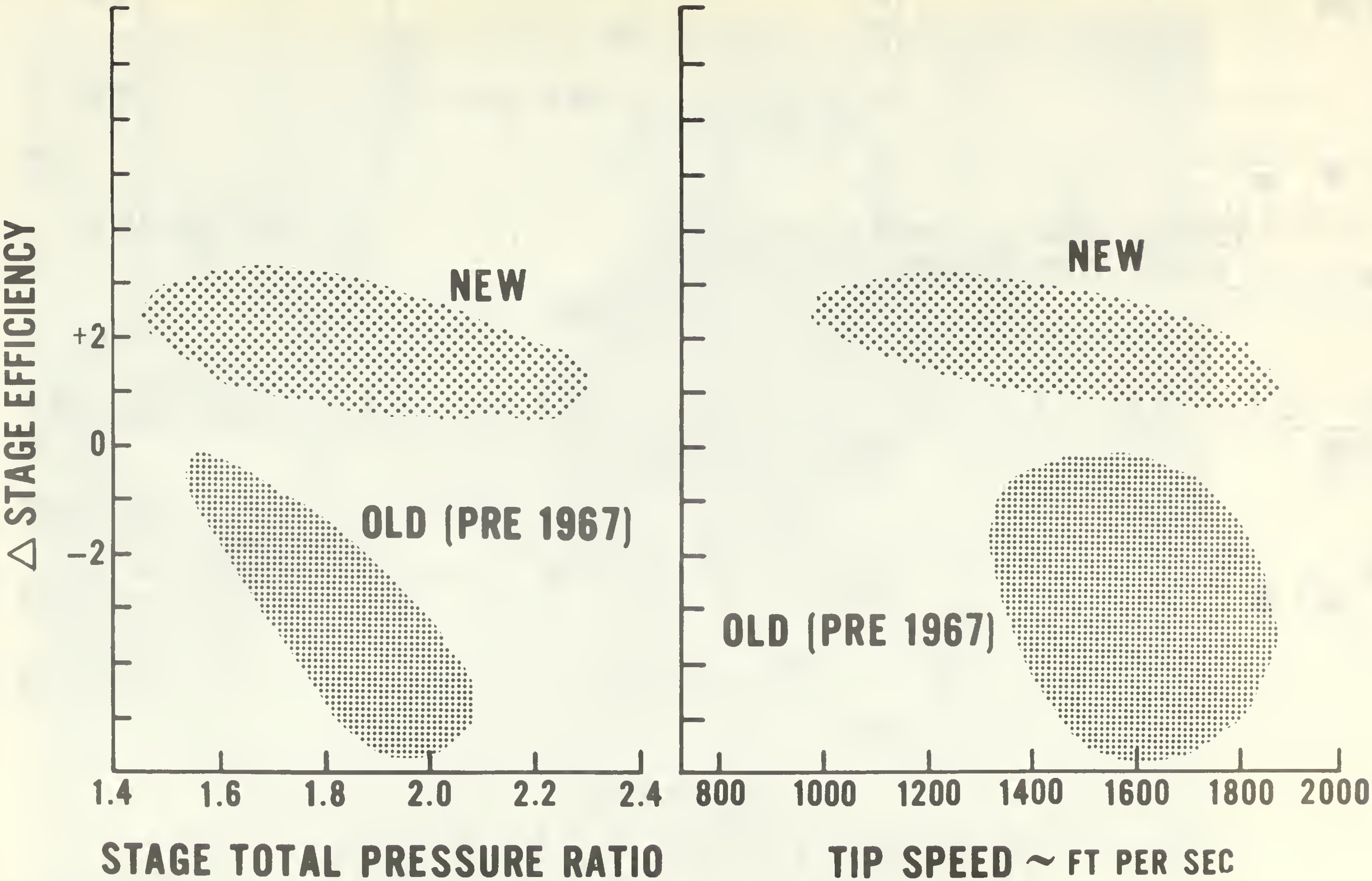


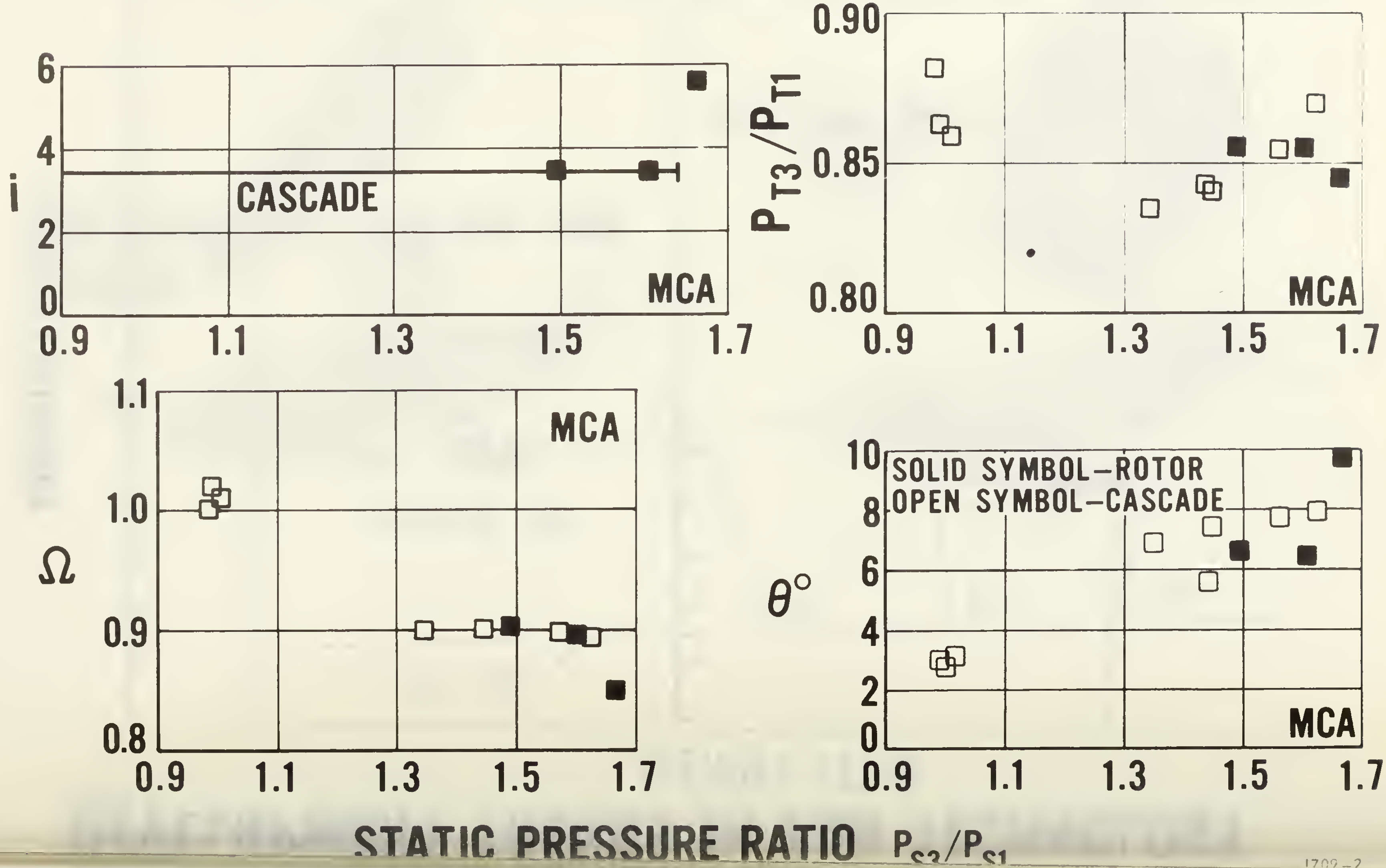
FIGURE 1. **DEVELOPMENT TRENDS IN FAN TECHNOLOGY**



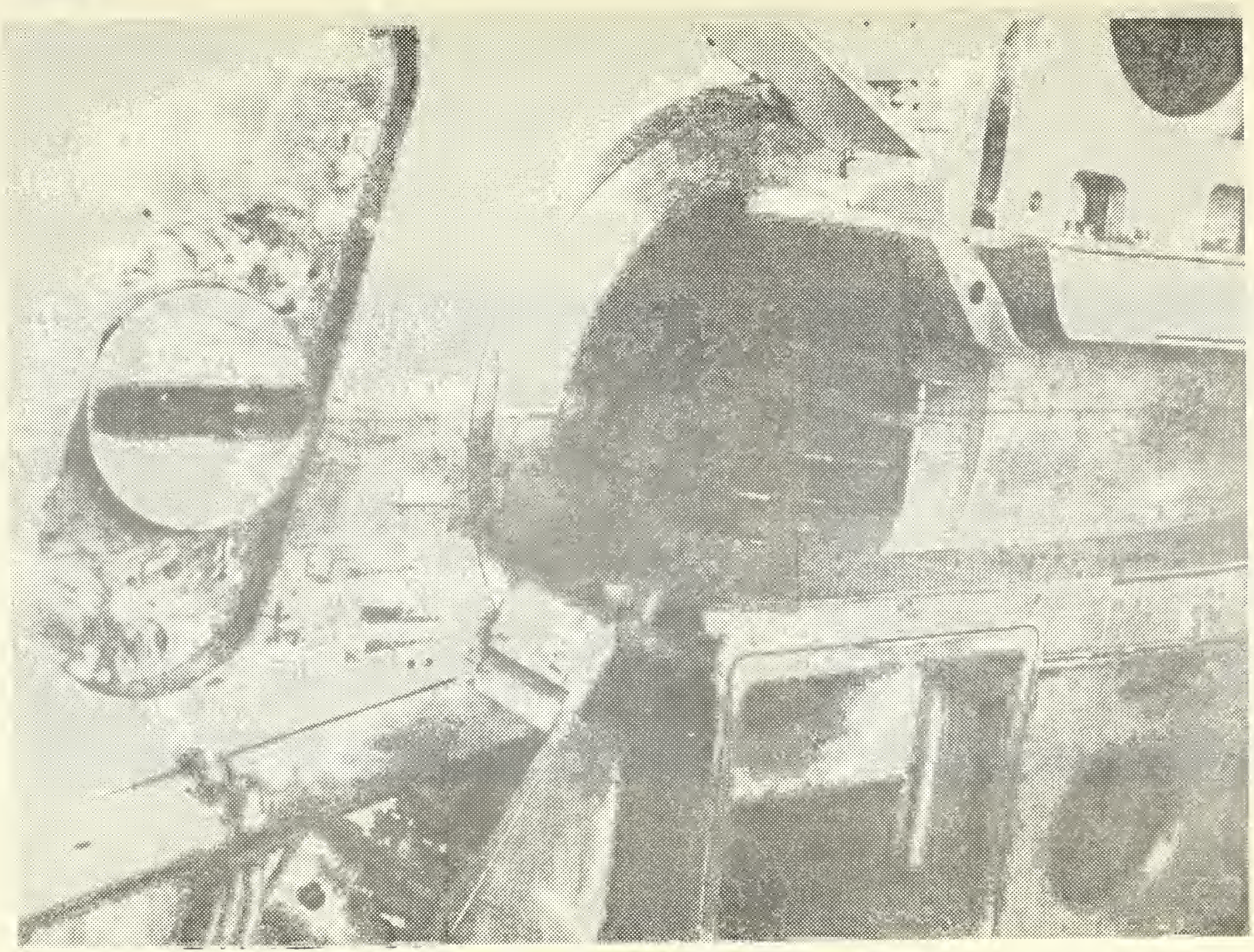
Pratt & Whitney Aircraft DIVISION OF UNITED AIRCRAFT CORPORATION

COMPARISON OF CASCADE AND ROTOR PERFORMANCE

MCA BLADE

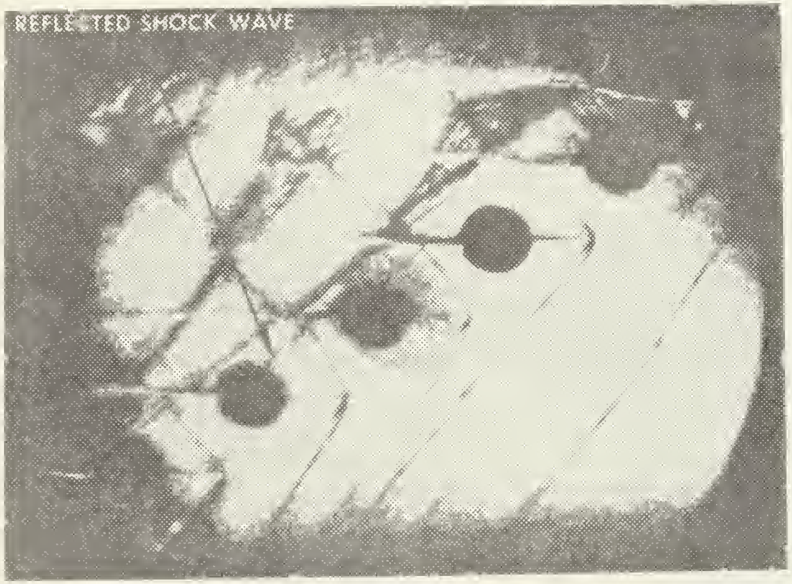


"TUNNEL TEST SECTION"

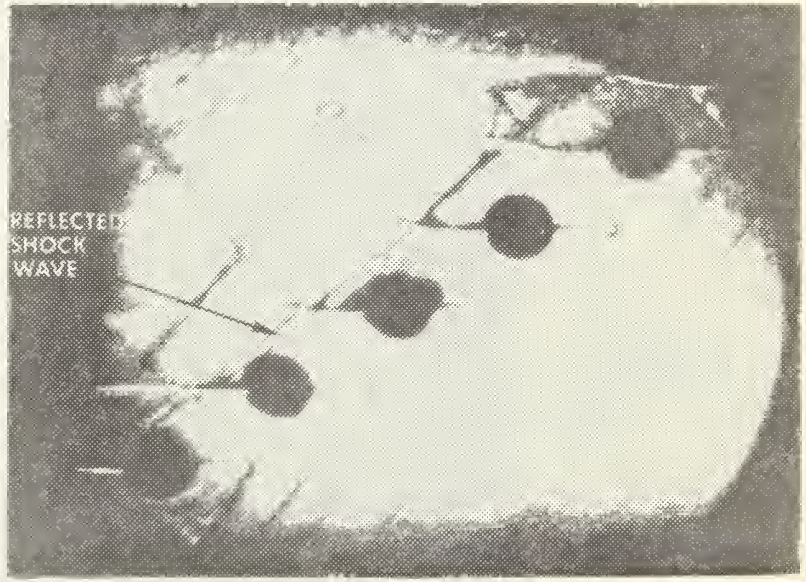


J709-12
702804

SCHLIEREN MOTION PICTURES SHOWING MOTION OF REFLECTED IN-PASSAGE SHOCK WAVE



t = 0



t = .001 sec.

Pratt & Whitney Aircraft DIVISION OF UNITED AIRCRAFT CORPORATION

NOISE

Discussion Leader: Dr. W. M. Foley

Introductory Remarks on Noise Research
by
W. M. Foley

Our chairman, Professor Vavra, asked me to say a few words on our research noise associated with supersonic tip speed fans. After reviewing the list attendees, however, I note that most of you are involved in other facets of rbomachinery research. I have, therefore, decided to broaden my remarks to ver the general field of noise research. I trust that this will be all right. ght off, I must admit that I am not certain just what the Navy's total noise oblem is. I suspect, however, that it is very closely related to the civil ise problem with which I am more familiar.

The first figure shows my assessment of three of the Navy's most important ise problems. The third one, community relations, is probably the most gnificant one from the standpoint of civil aviation, but it may not be of most portance to the Navy. It would, however, be very important to the Commanders Naval Bases. The first and second items would, of course, be important both sea and ashore.

The entire noise problem is a very complex one. The second figure attempts show some of its important facets and how these all interact and have an effect on the noise problem. Since this meeting is confined particularly to turbomachinery, certainly do not wish to bore you with discussions on community relations, airport erations, etc. The lower category shown in the figure that of aircraft design, certainly is an appropriate area of discussion at this meeting, however. Thus, I will concentrate my remarks on the effect of engine design parameters on aircraft noise.

ENGINE NOISE SOURCES

Figure 3 attempts to illustrate the sources of noise from an engine. As you can see, all major components of the engine contribute to the noise. The jet exhaust issuing from the engine also is a major source of noise. Further, the noise from the various sources can be broken down into several categories; broadband noise, discrete tones, and combination tones. We will look at each of these classes of noise in turn, and perhaps the meaning of each category will be well understood before I complete the discussion.

Examination of Figure 4 shows immediately the various sources of noise being discussed. Focus attention only on the upper portion. In the far left is a region entitled "Jet Noise." Note that this is low frequency noise with no particularly discrete peaks, and it is broadly distributed over a wide range of frequencies below 2000 cycles. In the middle of the slide are several discrete tones labelled as "Fundamental" and "Harmonics." These are created by the rotating blades and are easily discernible as being at the blade-passing frequency at which the blades are moving past an observer and past stationary objects in the engine at harmonics of these blade-passing frequencies. All rotating components of the compressor and the turbine create fundamental tones; however, those from components in the front and rear of the engine are most discernible. At a somewhat lower level, but still quite intense, is a broad range of noise above the jet spectrum. This has been called broadband noise and arises from many sources; the interaction of turbulence with the moving blades, so-called "vortex shedding" from the moving blades, and numerous other sources which are not well cataloged at the present time. Finally, between the jet noise region and the region marked "Fundamental Tones" there is a region marked "Combination Tones." This is a region which has discrete frequencies in it but the tones are all at frequencies less than the blade-passing frequency. When these combination tones were originally observed, they created quite a stir because one expects to see the fundamental and various higher harmonics but no lower harmonics. This phenomenon arises particularly in fans with supersonic tip speeds, and we will look at the mechanism for its generation in more detail later in the talk.

The data in Figure 4 were taken in front of the engine and represent the forward-radiated noise. Had we placed a microphone to the rear of the engine, a similar -- but somewhat different -- spectrum would have been observed. Also, this spectrum would vary with throttle setting and, of course, with engine design parameters.

An observer on the ground who hears an aircraft fly over is exposed at one time or another to all the sources of noise within the engine. This fact is illustrated in Figure 5 in which it is shown that as the aircraft approaches the observer the inlet and compressor noise are heard most prominently. When the aircraft gets overhead, the fan duct discharge noise and the jet noise tend to be the dominant sources. Finally, as the aircraft gets well past the observer so that he hears the aft-radiated noise, the low frequency jet noise is the dominant source. Thus, when one is concerned with minimizing the effect of noise on personnel on the ground he must be concerned with all noise sources within the engine.

JET NOISE RESEARCH

There has been considerable noise research in progress for over ten years. A significant portion of this research has been concerned with the suppression of noise or the treatment of the engine structure to prevent noise radiation. I wish to touch on this subject only very briefly today, since there are undoubtedly others in the audience more qualified to speak on this subject. Another area which has received very significant emphasis is psychoacoustics. While this is a very important area in the overall noise problem, it does not seem appropriate for this meeting, so I will not address myself to that problem area at present.

The subject of jet noise research has undoubtedly received the most emphasis of the several noise areas appropriate to today's meeting. This work reached its peak at the time that commercial jet airplanes were being introduced into service. It then decreased in activity for some period of time and is again reaching a peak because of the noise associated with the engines for the Supersonic Transport. It is well known that at intermediate and high subsonic jet velocities, the noise increases as the eighth power of the jet velocity. At supersonic speeds the increase is at a somewhat lesser rate. Because of the high jet velocity associated with most military aircraft, jet noise tends to be the dominant noise source on takeoff. Thus, if one is concerned with the noise problem of military aircraft, one might first reduce the jet velocity. This tactic has already been employed to significantly reduce noise levels of current civil aircraft. The results have often been strange and confusing, however. At speeds below about 1000 fps, jet noise results generally cease following an eighth power law and have often followed laws which vary with the particular investigator. In Figure 6 I have selected some recent results which have been taken in an attempt to ascertain why noise data obtained in various places led to different results at speeds below 1000 fps. We find that if one is particularly careful to eliminate extraneous noise sources so that only the true noise due to the external jet is heard, the noise follows the expected eighth power law down to very low speeds, say 300 fps, as shown in the figure. However, if one is not cautious in eliminating upstream noise, other sources creep in; for instance, just the simple pipe, Curve C, leads to higher noise at speeds below 600 fps.

Insertion of a combustion chamber, even though no combustion is initiated, increases the noise to the level shown by Curve B. This actually exceeds the noise of a turbojet engine, Curve A. These data serve as a warning to investigators who are working at low jet velocities in an attempt to use increased bypass ratio to reduce jet noise. Great care must be exercised in obtaining meaningful measurements.

Once results such as those shown in Figure 6 are well understood, it becomes clear that jet noise can be readily reduced by decreasing the jet velocity. However, one quickly reaches a noise floor set by other noise sources. On the basis of noise results, such as those shown in this slide, a clear understanding of subsonic jet noise has resulted. While I am sure that some would disagree with me, it is my personal feeling that subsonic noise from single jets is now adequately characterized for engine design purposes.

What happens when for other reasons we cannot reduce the jet velocity to obtain a mandatory noise decrease? Is there some way that noise can be suppressed? Figure 7 illustrates results obtained at Pratt & Whitney using noise suppression devices. The upper line shows the noise resulting from a simple jet at velocities greater than those shown in the preceding slide. Note that at velocities over 2000 fps there is a change in the slope of this line such that the noise no longer follows an eighth power law as it does at lower velocities. The lowest line on this figure illustrates the reduction in noise which was achieved by use of suppression techniques; as much as 10 decibels could easily be removed from the jet sound pressure level. The sound suppressors, however, resulted in a changed noise spectra. In determining the influence of noise suppression techniques on the annoyance of personnel on the ground, this changed spectra must be taken into account. The intermediate line, shown on the figure, shows the result of calculating the perceived noise; that is, the effective noisiness insofar as personnel are concerned. Note that at jet velocities up to 1500 fps very little decrease in annoyance was achieved even though significant total sound power level reductions were achieved. At the highest velocities employed, however, a significant perceived noise reduction was achieved due to suppression. One concludes from such results that suppression may be useful for noise alleviation in some military aircraft and the SST; for normal subsonic jets, particularly transports, sound suppression in the jet is not particularly effective because the jet velocities are well under 2000 fps.

Noise suppressor test results obtained by the Boeing Company are shown in Figure 8 for a pressure ratio 3 nozzle. This particular configuration was chosen to illustrate what can be done for an engine with an afterburner. We can see that under these operating conditions a very significant sound pressure level reduction is achieved by the addition of a 37-tube sound suppressor with corrugated ends. The addition of a liner surrounding each of these small tubes reduced the noise further so that with the complete suppressor approximately 15 decibels were removed at most frequencies. Since this device was effective over a wide range of frequencies, the perceived noise level was also significantly reduced. Considerable work needs still to be done in understanding the mechanisms that led to this noise reduction and also in understanding the tradeoffs between nozzle suppressor design and performance. For instance, in the example shown in Figure 8, there was a reduction in thrust of 8 percent, which itself is quite significant and would mean that an engine fitted with such a suppressor would have to be made oversize. Sound suppression on afterburning turbojet engines is a problem area which requires continued research.

We have already seen how reducing the jet velocity decreases the noise. One method of creating an efficient engine for subsonic applications while taking advantage of the noise reduction resulting from reduced jet velocity is to use a high bypass ratio fan engine. Figure 9 illustrates how noise can be reduced at constant thrust by varying the velocity between the secondary jet or fan air, U_2, and that of the gas generator or primary jet, U_1. There is an optimum velocity ratio near one which leads to the maximum attenuation. The upper curve in the figure is for an area ratio of 1.2 between the primary and secondary nozzles and the lower curve is for an area ratio of 1.93. Note, then, that not only the velocity ratio but also the area ratio is an important parameter. Investigations of a large number of jet nozzles with different bypass ratios have allowed the optimum bypass ratio to be selected for particular installations. This fact is illustrated in Fig. 10. For an engine like the JT9D with a bypass ratio of 5 something over 30 decibels noise reduction results over that which would be present if a pure turbojet engine were used for the same purpose. It is my current feeling that the effect of bypass ratio on jet noise is well understood but that employment of high bypass ratios to reduce noise leads to a number of other problems which are not well understood, since the noise floor for the engine is then set by sources other than the jet.

FAN NOISE RESEARCH

The selection of high bypass ratio to reduce noise still leaves us with the problem of obtaining good efficiency and low weight in an engine design. These constraints result in a requirement for high fan tip speeds. Thus, it is appropriate to now examine noise from engine fans. Figure 11 illustrates the noise intensity resulting from several types of noise within a fan at various tip speeds. At low tip speeds, such as would be employed during approach, the broadband noise resulting from many poorly understood internal noise sources is dominant. At intermediate speeds, the blade passage frequency noise resulting from interaction of wakes from the upstream fans with downstream stators and also the noise from such interactions in the compressor become dominant. Finally, at the very highest tip speeds, where the fan is moving with supersonic tip speeds, the so-called multiple pure tones become dominant. One must, therefore, be concerned with reducing all of these noise sources if noise is to be reduced over the entire operating range of the engine. Let us re-examine some typical engine spectra which show multiple pure tones. In the upper graph of Figure 12, spectra taken from a small 4-in. diameter scale engine are shown. Note that discrete tones at less than blade-passing frequency are clearly discernible; in fact, the intensity of a number of these tones is considerably greater than that of the primary fan blade-passing frequency. In the lower graph is a spectra taken from the JT9D engine. Because of the different rpm and numbers of blades between the 4-in. model and the almost 8-ft. engine the frequencies are different, but many discrete tones at less than blade-passing frequency are clearly discernible. Again, some of these tones are considerably more intense than that of the primary fan blade-passing frequency. The general character of the spectra of these two engines differing in size by a factor of almost 25 is very similar.

As I mentioned previously, the existence of multiple pure tones was a considerable surprise when first observed. One would expect that the pressure pulse from the shock wave from each blade would lead to intense sound at the blade-passing frequency and at higher harmonics. The mechanism to generate such intense frequencies at lower than blade-passing frequency was not initially obvious, however. The observance of multiple pure tones and the fact that these were so intense has led to significant research programs on this phenomenon at the several major engine manufacturers. One of the facilities which we have used to study such problems is illustrated in Fig. 13. This is a supersonic cascade tunnel, and it has been used to probe the flow fields ahead of the blades in the region where the shock waves from the several blades are interacting and to investigate the effects of such parameters as blade alignment and leading-edge radius on shock wave intensity. Similarly, on both a small-scale and a large-scale rotating fan rig, probes have been inserted at varying distances upstream of the rotor to investigate the rate of attenuation of the shock waves propagating from each of the blades. Such experimental studies have been paralleled with detailed analytical studies, and a good explanation for multiple pure tones is now in hand. Each fan blade generates a shock wave whic is followed by an expansion fan. Depending upon the blade alignment, thickness, leading-edge radius and other physical parameters, the intensity of the shock wave created by each blade will differ. This results in the shock waves moving upstream at varying velocities, some overtaking others with the resultant variations in spacing and intensity of the waves. Hence, frequencies as low as one per revolution of the fan are discernible in these patterns. The prediction of these tones must be treated as a nonlinear aerodynamics problem. It is observed that very small alignment or thickness errors have a very noticeable effect upon the noise and that significantly larger errors do not produce significantly larger increases in multiple pure tone noise. Thus, it is virtually impossible to produce a fan which is sufficiently perfect that it does not produce such tones. In practice, even if such a fan were produced, it would not remain so for long due to erosion, foreign object damage, etc. It does not, therefore, appear feasible to eliminate combination tone noise by careful blade selection or tight manufacturing tolerances. Proper selection, however, of suitable material to attenuate the waves in the vicinity of the blade tips before the waves have an opportunity to spread out differentially and lead to combination tone noise appears to be a feasible way of reducing this phenomenon. One such example is shown in Fig. 14. Further research is required on just how best to accomplish this.

We saw in a preceding figure that blade-passing frequency noise was dominant at some fan speeds. A primary source of this noise is the interaction of moving blade wakes with downstream stationary blades and vice versa. The choice of blade-vane spacing can produce quite an effect upon the generated noise. This fact is illustrated in Figure 14. Although the data for this figure were obtained in an engine with upstream inlet guide vanes, a similar result is obtained by selection of proper spacing between fan blades and their stators. The modified engine was produced by moving the upstream guide vanes, which were originally placed very close to the blades, a distance of two chords upstream. Note how this effectively eliminated the harmonics of the first blade-passing frequency and reduced the primary blade-passing frequency by a full 10 decibels. The effect of blade-vane spacing is now fairly well understood qualitatively; however, insufficient data are in hand to let the designer select what noise benefits he can expect from a particular spacing selection. Thus, further work is still required to provide adequate design data.

At low fan rpm's we noted previously that a dominant noise source was that entitled "Broadband Noise." This noise results from several sources; turbulence interaction with the blading, boundary layers, nonuniform lift forces on the blades, and vortex shedding. This latter phenomenon, vortex shedding, is hypothesized to result from the occurrence of a vortex wake similar to that shed from a cylinder, but less coherent. Most of the sources of broad-band noise are, at present, very poorly understood. Benzakein and Hochheiser from General Electric attempted to determine the relative intensities of the several broadband noise sources known to exist in fans in a paper presented at the 1970 ASME Annual Meeting in New York City. Their results are depicted on Figure 15. Note that the so-called vortex shedding noise from the fan blades is the dominant source followed by the vortex shedding noise from the outlet guide vanes. Turbulence interactions appear to be much less important. Our own studies would tend to confirm the General Electric results. It is expected that such broadband noise sources will set the floor level of noise in future engines once combination tone noise is eliminated. For this reason, it is my feeling that broadband noise deserves significant attention in order to understand it in sufficient depth so that it can be reduced to a minimum.

INSTALLATION EFFECTS

There must, of course, be almost an infinite variety of installation effects -- some of the major; for example, such things as panels which are excited by engine vibration can be significant noise sources. Today, I would like to emphasize just one major installation effect which has recently become known since this could have an important effect upon jet noise. I am sure that many of you are already aware of the so-called "spinning mode" theory of Tyler and Sofrin which was developed about a decade ago. By use of this theory it is possible to pick the number of rotor and stator blades such that the interaction noise at blade-passing frequencies and higher harmonics can be cut off and will not propagate. Since this theory is well understood, I do not wish to discuss it here. However, in the engine installations of the B70 and the F-111, a new cutoff phenomenon has been observed which is not present in commercial aircraft installations with short inlet ducts. This cutoff does not appear to be due to wall treatment. The upper curve in Figure 16 shows how the sound pressure level increases with engine rpm in a typical short duct installation. However, on the B70 the first blade-passing frequency intensity falls rapidly as rpm increases. A similar phenomenon has been observed in the F-111. It is well known that the long duct forces the sound waves to spiral many times around before propagating from the duct. This leads to significant attenuation. This is particularly true when the inlet velocities in the duct are near Mach 1.0 so that the wave angle is very slight. This effect does not appear to be sufficient to account for the attenuation which has been observed in these installations, however. The problem of predicting the noise propagation from a long duct with a realistic velocity distribution and wall characteristics has not yet been solved. In view of these recent results, it appears that duct propagation is another problem which deserves continued attention.

SUMMARY

I have discussed a number of engine noise sources which may be of interest in Navy installations and which are deserving of further attention. These problems are those of jet noise suppression in high velocity jets, characterization of broadband noise in order to learn how to reduce or suppress this noise phenomenon, and the attenuation in long inlet ducts. We at United Aircraft are sufficiently concerned about these several noise problems that we have just completed a new wind tunnel specifically for noise research. This tunnel is currently being calibrated aerodynamically and acoustically, and is shown in Figure 17. Basically, it is a free-jet wind tunnel with an open test section surrounded by an anechoic chamber. Objects which we wish to study can be immersed in the airstream, and the sound is then allowed to propagate out across the jet boundaries to microphones in the far field which can measure the noise. Engine components, such as scaled fans, exhaust nozzles, and compressors, can be run within the airstream.

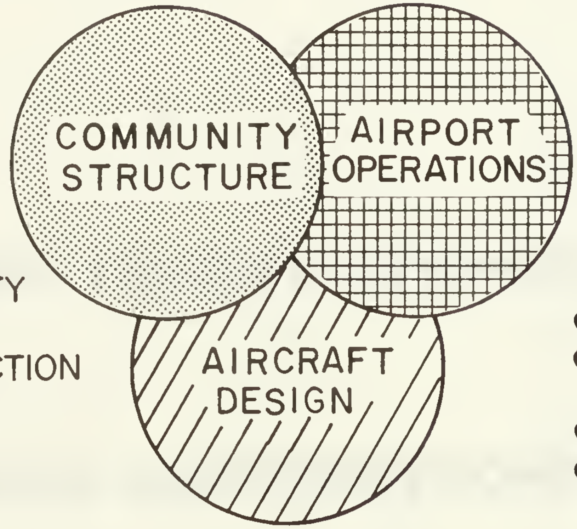
HIGH NOISE LEVELS

- ARE DETRIMENTAL TO FLIGHT AND GROUND CREWS
- RESULT IN A POOR COMMUNICATIONS ENVIRONMENT
- ARE DETRIMENTAL TO NAVY/COMMUNITY RELATIONS

FIGURE 1

United Aircraft
Research Laboratories
UNITED AIRCRAFT CORPORATION

STRUCTURE OF NOISE PROBLEM



- POPULATION DENSITY
- LAND USE, ZONING
- BUILDING CONSTRUCTION
- HUMAN REACTION TO NOISE

- USAGE:
  - TYPES OF AIRCRAFT
  - FLIGHT RANGES
  - DENSITY OF FLIGHTS
- FLIGHT PATTERNS
- CONTROL OF FLIGHT PROFILES
- BUFFER ZONES
- ASSISTED POWER TAKE-OFF

- MISSION REQUIREMENTS
- CLIMB AND LANDING CHARACTERISTICS
- NACELLE LOCATION
- INLET DESIGN
- EXHAUST CONFIGURATION
- SOUND-ABSORBING LININGS
- POWERPLANT SELECTION
  - THRUST
  - NUMBER
  - CYCLE
- POWERPLANT DESIGN DETAILS

FIGURE 2 *AIRCRAFT-AIRPORT-COMMUNITY SYSTEM*

ENGINE FAR FIELD NOISE IS COMPOSED

OF NOISES FROM SEVERAL SOURCES

- PRIMARY JET NOISE
- FAN DISCHARGE JET NOISE
- INLET FAN NOISE

  BROADBAND
  TONES
  COMBINATION TONE

- DISCHARGE DUCT FAN NOISE

  BROADBAND
  TONES

- TURBINE NOISE

  BROADBAND
  TONES

- ENGINE COMPRESSOR

  BROADBAND
  TONES

FIGURE 3

AIRCRAFT ENGINE NOISE SPECTRAL COMPONENTS

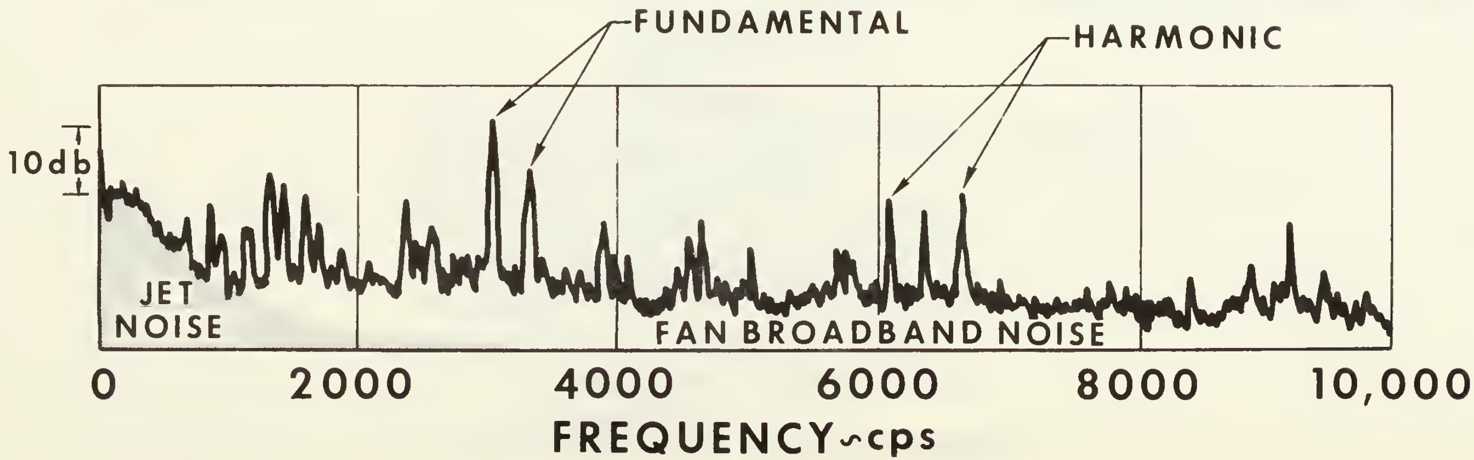
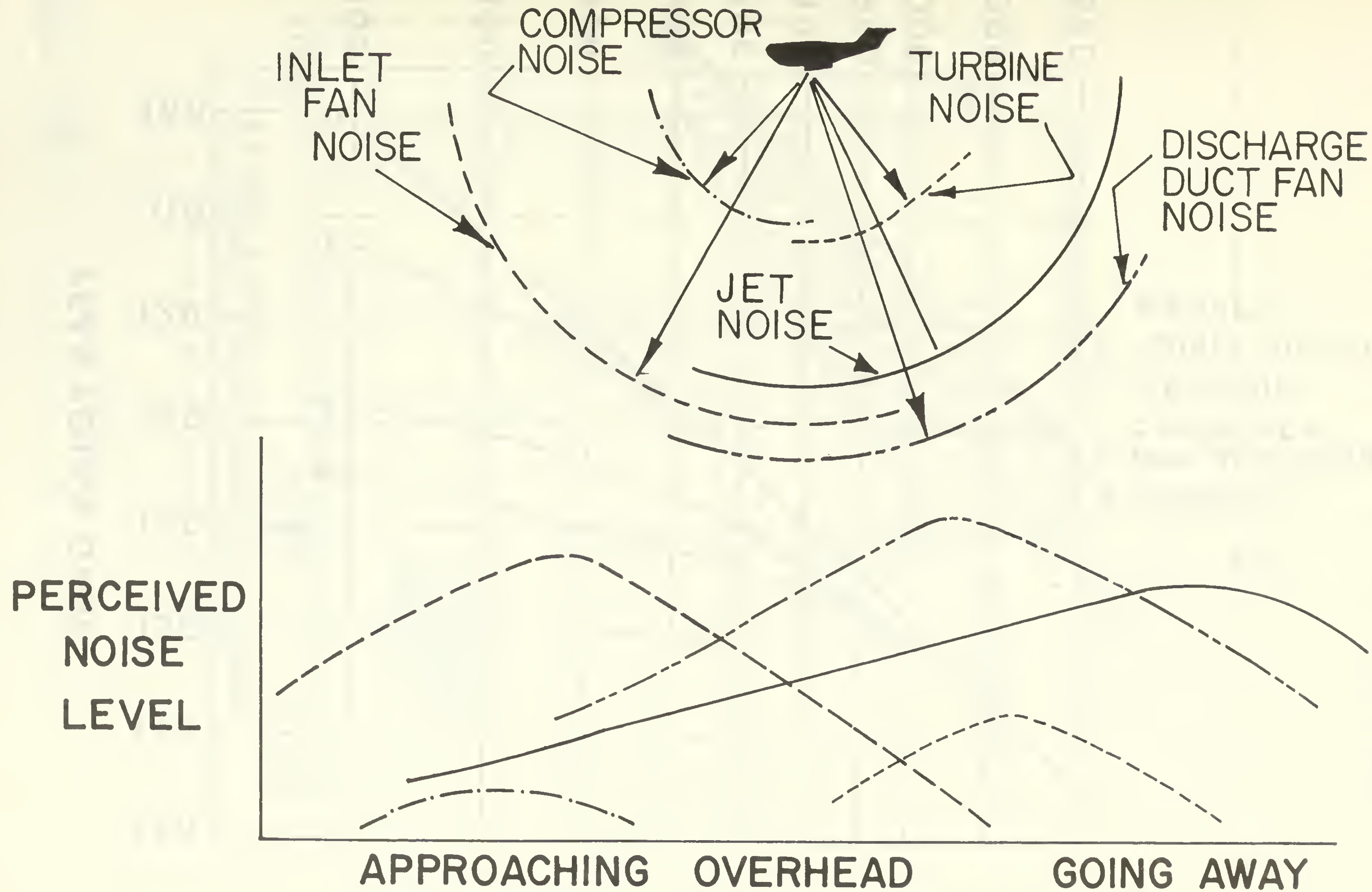


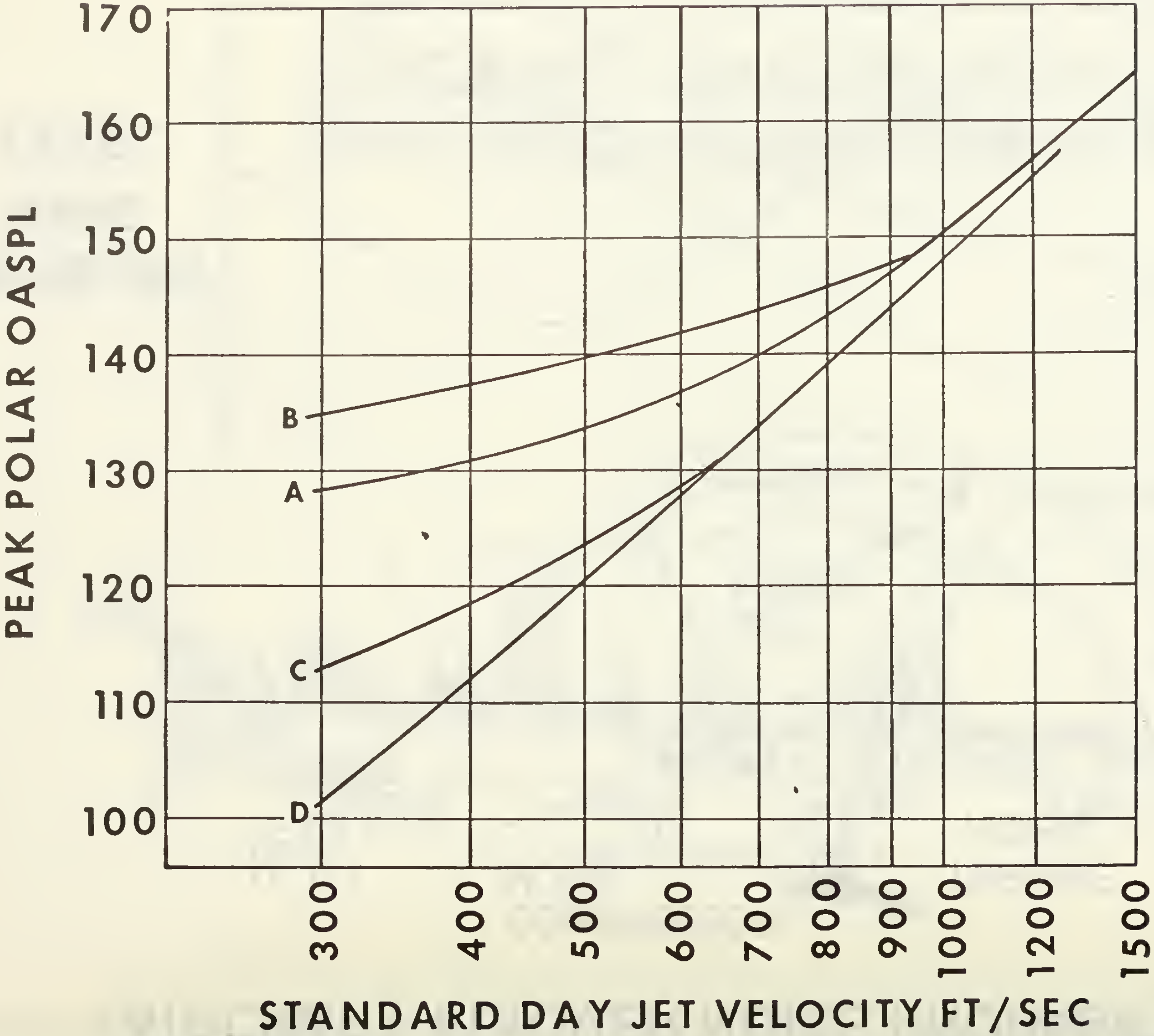
FIGURE 4

United Aircraft
Research Laboratories

AIRCRAFT FLYOVER NOISE PROPAGATION



JET NOISE CORRELATION



KEY:

A. TURBO JET
B. PIPE WITH UPSTREAM COMBUSTOR
C. SIMPLE PIPE
D. "QUIET" UPSTREAM AIR SUPPLY

United Aircraft
Research Laboratories

JET NOISE SUPPRESSION

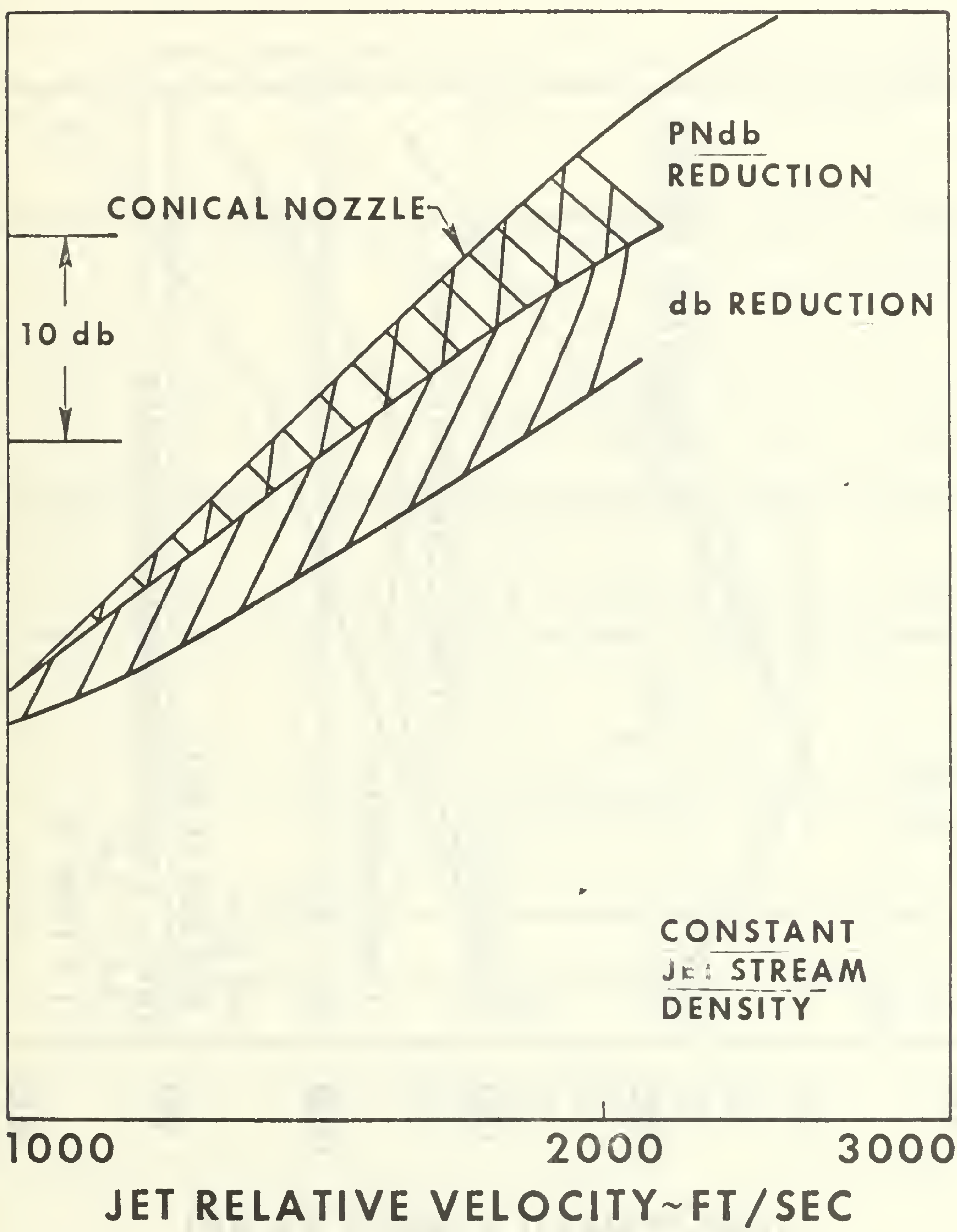
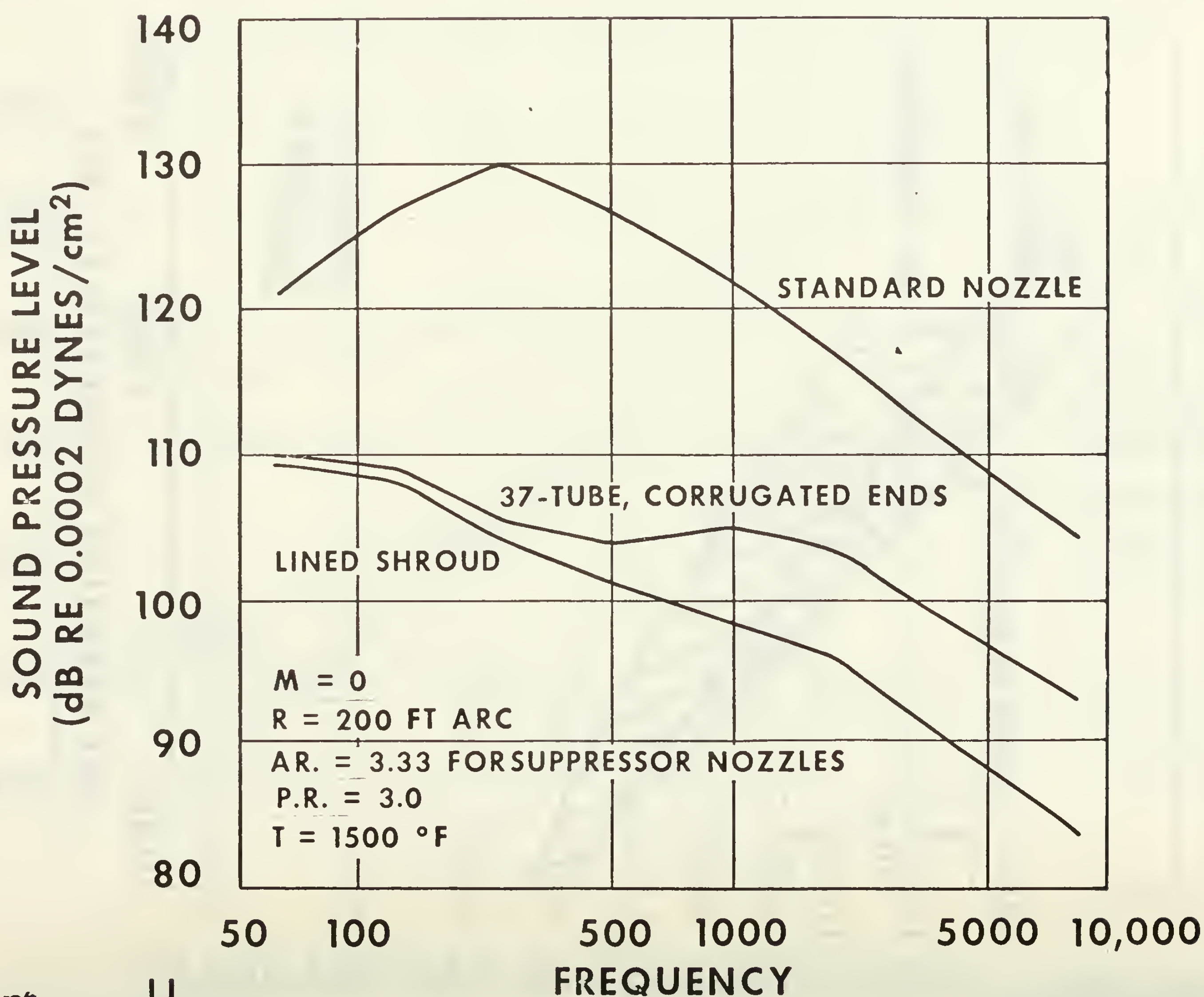


FIGURE 7

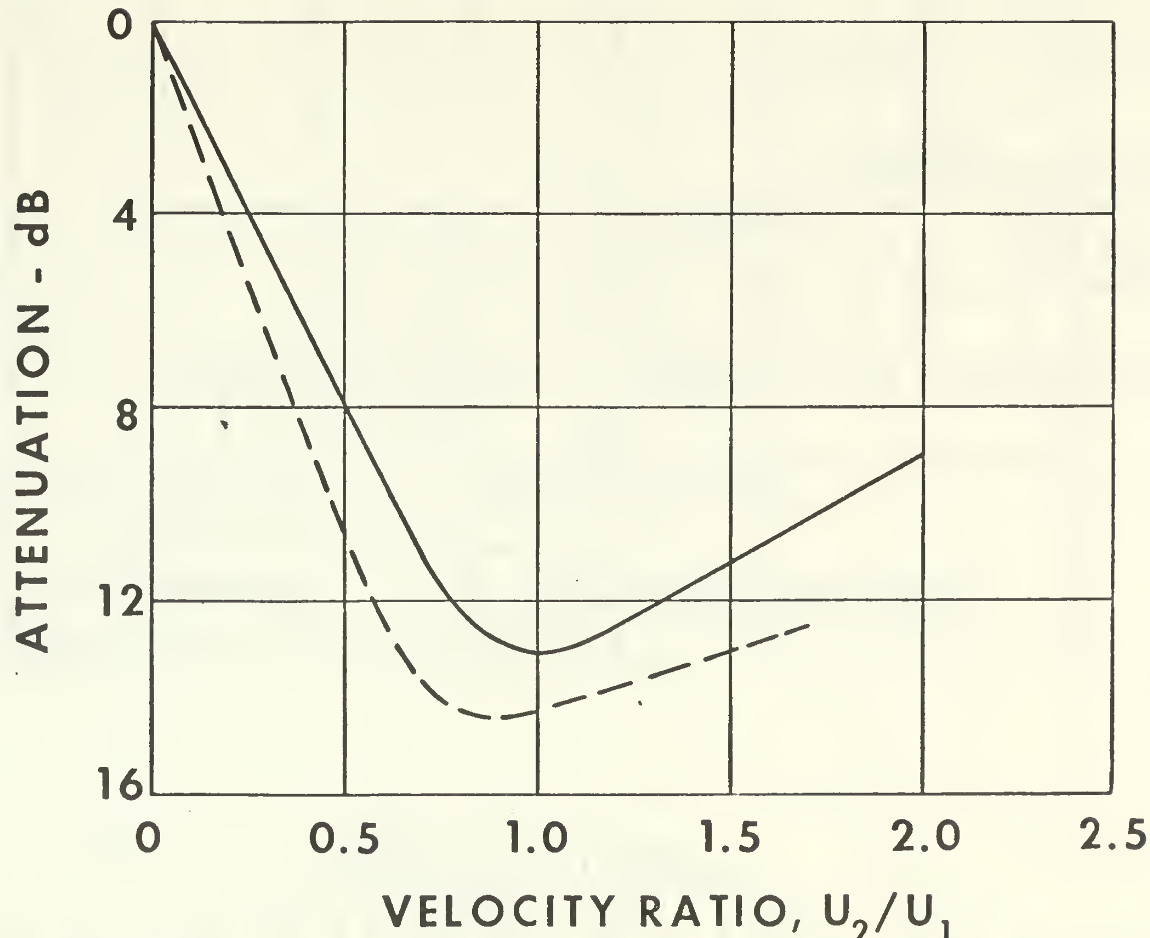
JET NOISE SUPPRESSION



United Aircraft
Research Laboratories
UNITED AIRCRAFT CORPORATION

FIGURE 8

ATTENUATION AS A FUNCTION OF VELOCITY RATIO FOR CONSTANT THRUST

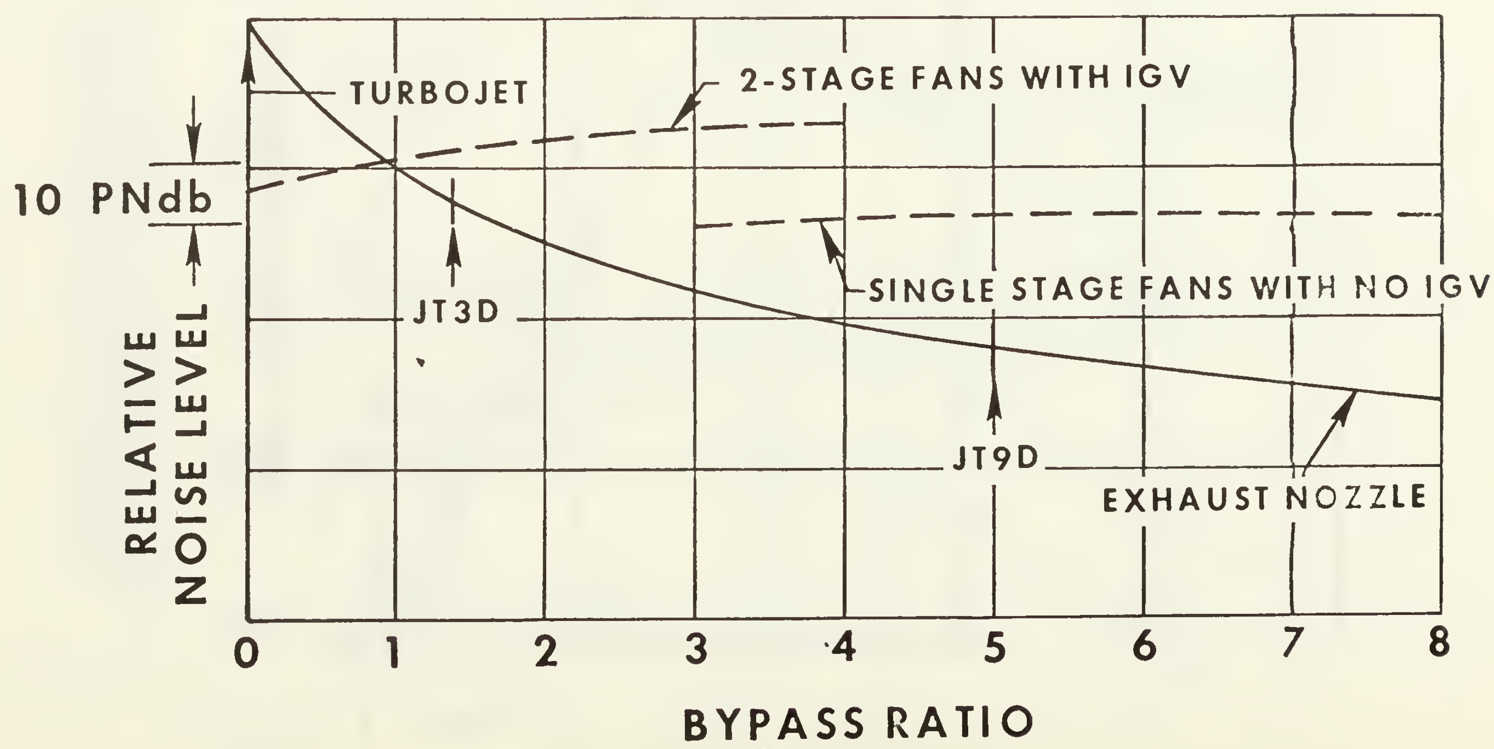


United Aircraft Research Laboratories

FIGURE 9

EFFECT OF BYPASS RATIO ON NOISE

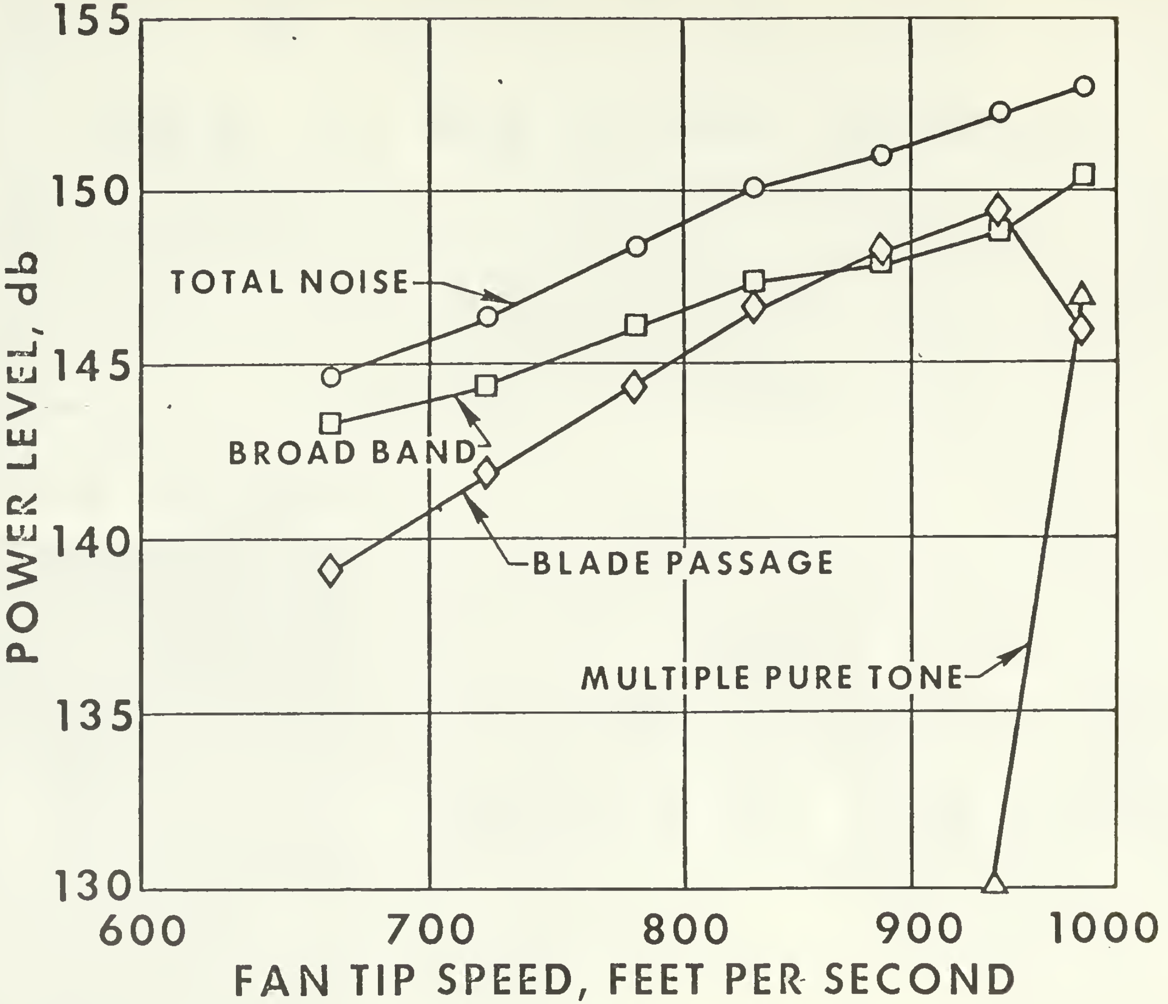
EQUAL TAKE-OFF THRUST



United Aircraft
Research Laboratories

FIGURE 10

EFFECT OF SPEED ON FORWARD-RADIATED NOISE



United Aircraft
Research Laboratories
UNITED AIRCRAFT CORPORATION
UA

FIGURE 11

FIGURE 6

EFFECT OF FAN DIAMETER ON COMBINATION TONE NOISE

FIGURE 6A JT9D AND SCALE MODEL POWERED NACELLE

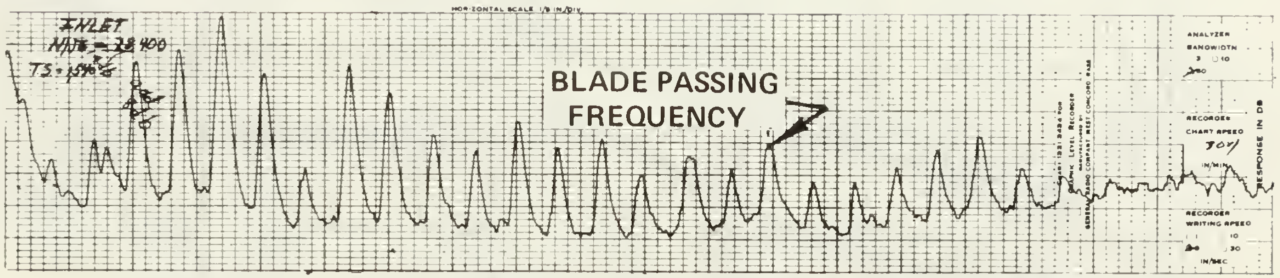


FIGURE 6B SCALE MODEL SPECTRUM 4 INCH DIAMETER

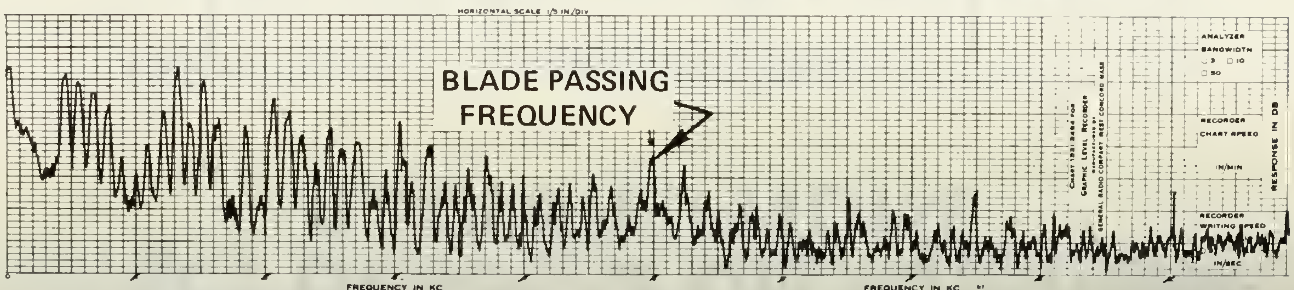


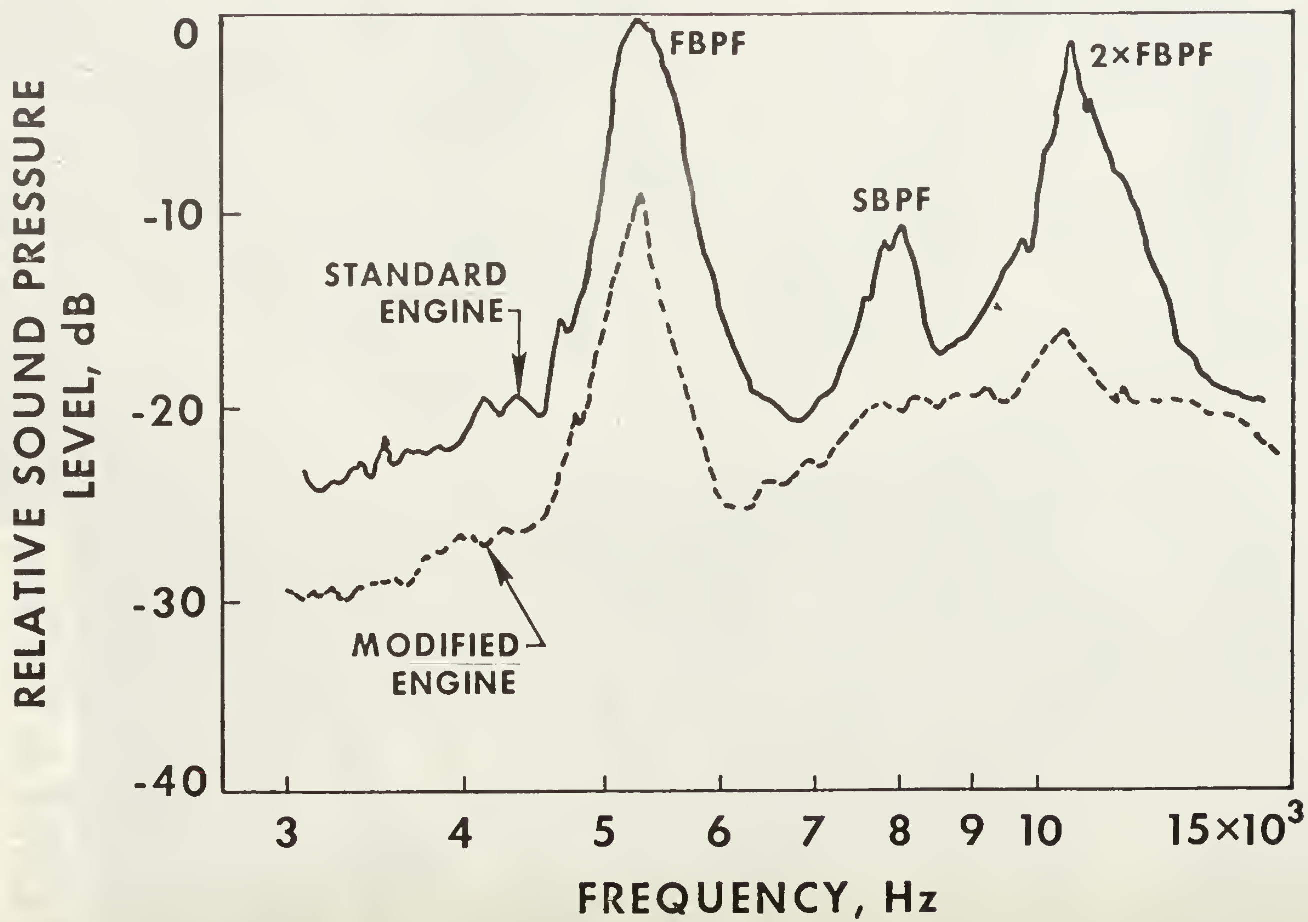
FIGURE 6C JT9D SPECTRUM 92 INCH DIAMETER

J1292-6
702508

IGURE 13

COMBINATION TONE NOISE FAN CASCADE RESEARCH

ONE-TENTH-OCTAVE BAND FREQUENCY ANALYSIS FOR STANDARD AND MODIFIED ENGINE FOR 30° AZIMUTH LOCATION



United Aircraft
Research Laboratories
UNITED AIRCRAFT CORPORATION

FIGURE 14

BROADBAND NOISE MECHANISMS ON A HIGH-SPEED FAN

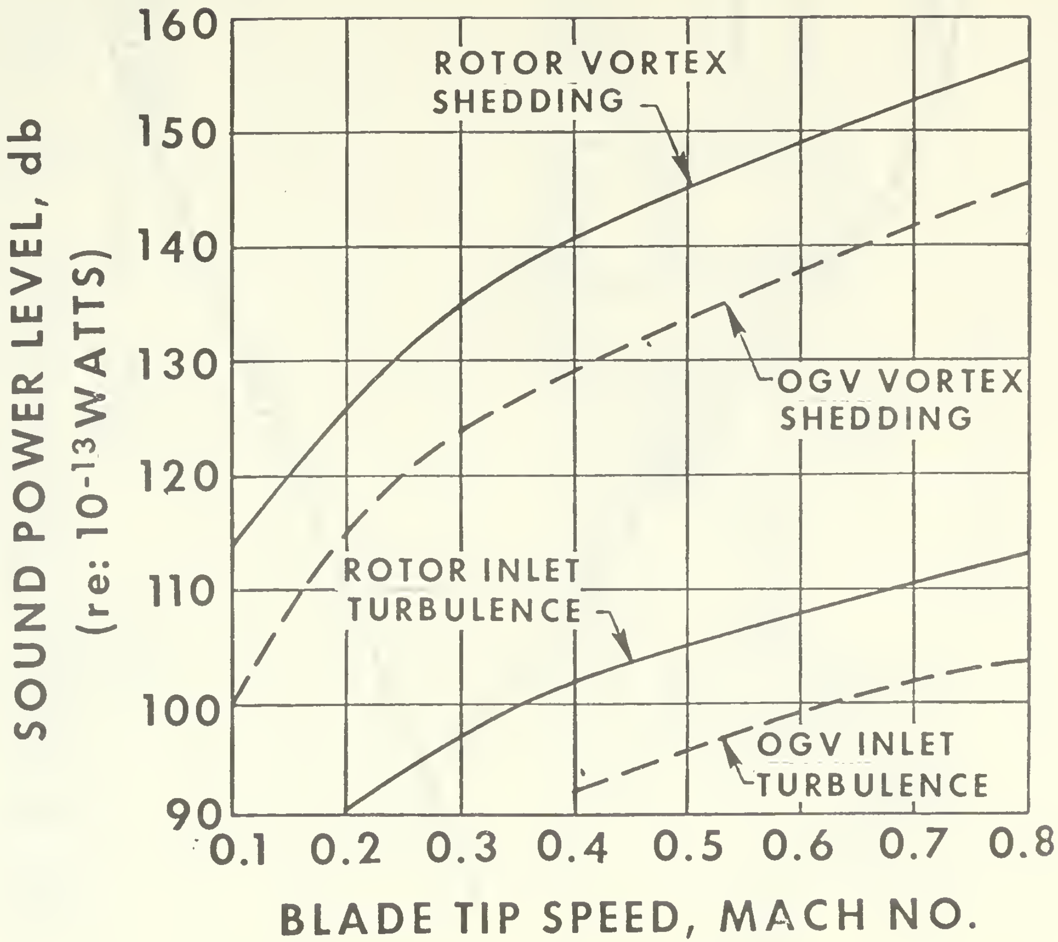


FIGURE 15

NOISE INTENSITY AT FIRST BLADE PASSING FREQUENCY

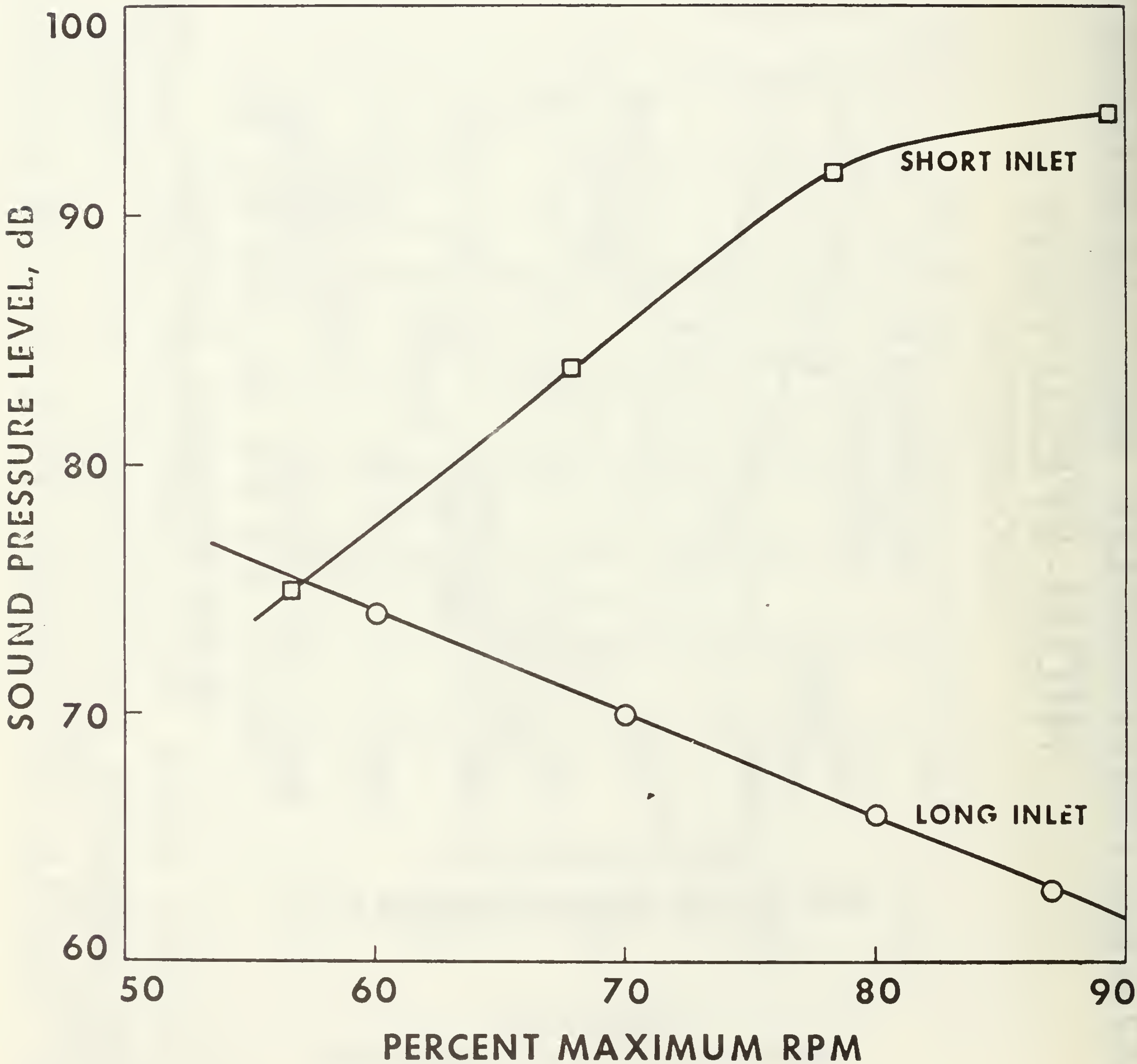
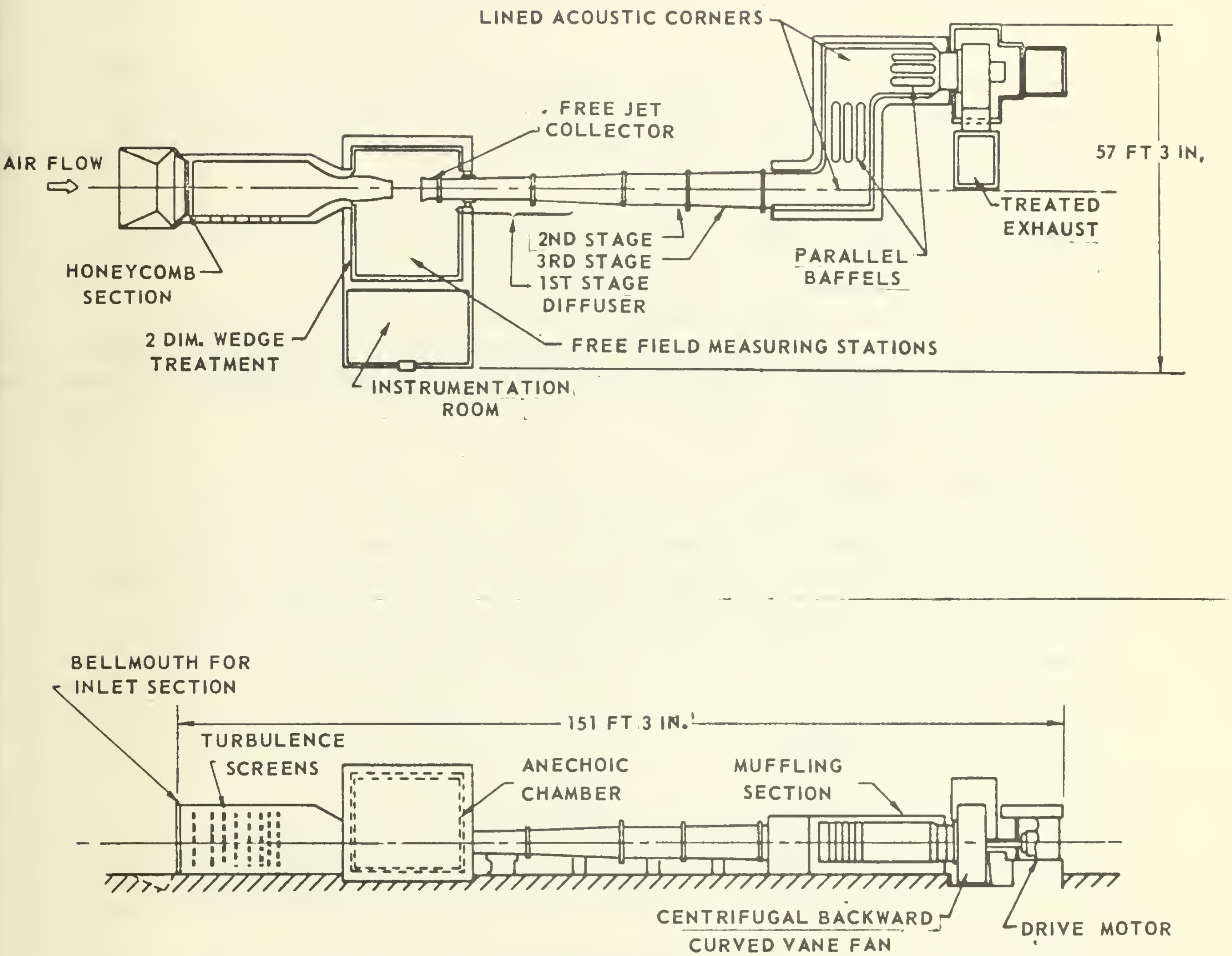


FIGURE 16

ACOUSTIC RESEARCH TUNNEL



PARTIAL SECTION A-A

FIGURE 17

DISCUSSION

(Yampolsky) * Where are your microphones located? Are the microphones far afield?

(Foley) The microphones are far afield forward, I've forgotten how far off axis, probably about 30 degrees off axis, and in the far field in both cases.

(Yampolsky)** Are those individual singular points or are they added?

(Foley) No, they are individual so they would be additive. But when you are talking about a decibel scale, the upper is obviously going to be dominant. I would expect it will set, in the near future, the floor for fan noise; so that is an area worthy of work. Some programs are in progress at the present time.

(Lakshminarayana) Can you briefly explain how you got the magnitude of various effects, how do you measure that?

(Foley) These were not measured. They were predicted, I believe, using simply the Sherland theory, as far as the vortex-shedding noise. I must admit I am not sure how the prediction for the turbulence interaction was. To my knowledge, there is not a real satisfactory theory available at the present time.

(Yampolsky) Whose paper is this from?

(Foley) I forget which paper, there were two given by the General Electric personnel at the ASME meeting last week in New York. I have the reference here if you want to look at it. It certainly agrees in general with our own predictions, and it did summarize it better than anything I had available so I chose to use it.

(Yampolsky) What sort of wave length in terms of the duct diameter are you talking about?

(Foley) Well, this is a first blade-passing frequency so it is probably a couple of thousand cycles. So we are talking about a wave length of a fraction of a foot in a duct that might be several feet. So the wave length is short relative to duct diameter.

(Yampolsky) But the far field is intermingled so that it is a confined radiation straight up, isn't it, a very confined beam?

(Foley) Right, but this is accounted for. In this case they were seeking the noise so they picked their microphone position to be near the maximum of the radiated sound.

*Editor's comment: This question refers to the first paragraph of page 10.
**Editor's comment: The three successive questions refer to the first paragraph of page 13.

(Yampolsky) One of the points is that, you mentioned it earlier, and that is the fact that there are still arguments of physics to explain what you are measuring. You can turn the system around, and your measurements may point to the model that might be used for calculation purposes, the three-dimensional effects in looking for wakes. I don't know what kind of resolution you can get from a microphone. Part of it is the normal scale that you use to record measurements and has too many orders of magnitude to it, that is, to the numbers we're used to. But I wonder if the general field of acoustics isn't the way we can get back at the three-dimensional problem and the periodicity problems inside the machine.

(Foley) Well, I don't know whether I have the total answer, but I might make a comment there. If one is concerned with very intense waves, like upstream of a supersonic rotor, then, of course, you can do very well with a transducer introduced in the flow field, and you will see the discrete shock waves going by your microphone or your transducer. But if you tried to poke some sort of a transducer down in the flow between the fifth and the sixth stage of some turbomachine, and the flow Mach numbers are reasonably high, or if you try to put a hot wire in there, you have a relatively complex problem. You have the stagnation of the unsteady flow against this transducer, and you have the acoustic waves incident upon it. They come from all directions, and it is quite difficult to sort one phenomenon from the other.

The hot wire works as a pretty good acoustic transducer. You may try to separate signals from the two mechanisms by inserting two wires, one downstream of the other. The acoustic wave, of course, will propogate at the speed of sound in the media, and the velocity fluctuations will pass at the mean speed. If you make your mean speed Mach number 0.5, you should be able to separate these and learn a great deal. We have, for example, tried acoustic sensors and hot wires and put them both in the same flow field. You find that life isn't simple. The waves are coming from all directions. They scatter, they reflect, and there is no simple way to sort the acoustic from the velocity fluctuations inside the machine. So, to my knowledge, using acoustic type sensors hasn't been a very fruitful method of probing flow internal to the rotor.

At lower Mach numbers hot wires are quite effective, but up at the Mach numbers that are present in a full scale machine they haven't been real successful.

(Serovy) I have been kind of interested over the last few years about what you would do to vary the angle of a rotor. Now, I know that we have variable stators, what if you could turn the rotor? Aerodynamically I think this has some very interesting possibilities. How would it be as far as sound is concerned? Isn't that a better deal? Couldn't you improve the noise characteristics?

(Foley) By varying the pitch of the rotor? Alex would you like to answer that?

(Mikolajczak) It is an interesting supposition. I have no comments.

(Hartmann) When upstream noise intensity is measured over a range of flow conditions at a fixed speed, we have observed cases where the maximum noise occurs near design flow. I don't know if this is the general effect nor do I know why this occurs. The variation from maximum to minimum flow is not generally large until the machine is stalled.

(Foley) In propellers we see this, where if you drive the propellers into stall, then the noise almost universally goes up. It doesn't take a very large region of stall on the blade to run the noise level way up. But the engine rotor is a little different story, I think, at least the supersonic rotor.

(Mikolajczak) I think what George was asking was not what happens as you go along a speed line, but what happens as we go down in speed toward approach conditions.

(Yampolsky) Well, you are talking about an engine and I think he is talking about a compressor. When you run a compressor in stall, your peak may not be as far; but you have a lot more energy in noise.

(Serovy) I am not a big engine man, so I don't know how these fans operate as you run along the engine operating line; but I have to assume that somewhere along the engine operating line the fan rotor is not doing very well for you. I've been interested in the possibilities of taking on -- I don't want to do it personally but you guys probably have to -- the horrible mechanical problems of a variable-angle rotor. I just wondered how that would hit you as far as noise is concerned?

(Mikolajczak) At part-speed in the subsonic portion of the fan, there could be a significant incidence change, compared to the design point operation; and restaggering could be significant for broadband noise generation. For the supersonic portion of the fan, if we consider combination tone for the moment, there is nothing to be gained by restaggering the rotor since the far-field Mach wave pattern should not be affected.

(Bullock) If a blade element is sufficiently "loaded", it will operate subcritically and stall, no matter what its designer had in mind. When the inlet flow is not uniform, we don't know the extent to which time-unsteady stalling may obtain.

(Smith) If we take the case of, let's say, a subsonic engine with a single-stage fan, like all of the modern high-bypass-ratio, subsonic engines, one of the noise conditions of greatest concern is at approach, at about 60% thrust or thereabouts. This is when the core of the engine is running still up pretty far in speed, but the fan is down at perhaps 60-70% of its design speed. Now at this condition, the flow relative to the blading may still be supersonic, but it is low supersonic. If this is a fan that was designed to have good cascades at its high supersonic design point, by high I mean maybe 1.4 Mach number, these blades are going to be operating at a high incidence angle along a normal fixed-exit nozzle operating line. As a fan engine, the incidence will be high. Now, if the flow is supersonic, it will expand around the leading edge. There will be a detached shock, the passage will not be started, there will be a strong shock that has moved not too far from the leading edge of the airfoil anymore, and this is the case where the leading edges of the blades are very heavily loaded. There is very little loading back toward the trailing edge. This is a rather noisy condition, because the airfoils would very much like to have more camber for this operating condition. Now, you can consider staggering rotor blades closed for such a case, in other words, closing them down. To get a better incidence condition you do, in fact, accomplish that. However, if it is still a flight requirement that you establish some percent thrust, such as 60% thrust which is a typical approach number, that means that the flow through the fan is going to be about the same regardless of how you stagger the pitch of the blades. So that when the blades are closed down to get a better incidence, then the fan will run faster. You are then in a trade situation between higher speed, better incidence on the one hand, and lower speed, poorer incidence on the other hand. And it is one of those things that isn't clear which way is the right way to go. I sense Alex would like to comment on my comment.

(Mikolajczak) I agree. I would like to mention that in some fans -- and this is not very well understood -- you find that as you go up the operating line to higher speeds, the combination-tone noise does not keep rising continuously. In some cases there may even be a maximum at part-speed. There are still areas in the noise generation that we do not quite understand.

(Smith) We certainly feel that way. When you show these curves that show noise going up with speed, you have to remember that these are results from a given fan run over a range of speeds; and it doesn't necessarily follow that if you have two fans designed for different speeds, that the high speed one is going to be noisier than the low speed one.

(Foley) Yes, I could have equally taken curves from the same paper with the fan at constant speed at three back pressure ratios; it was the noisiest at the intermediate back pressure ratio and low in noise at a high back pressure and a low back pressure.

(Smith) Are you sure you don't have that backwards? I know of results which are just the other way.

(Foley) We have seen results like that too. I am saying that that was a particular result for a particular fan. It was at a low tip speed, so it was just barely transonic; and this may be one of the reasons that it lead to some rather surprising noise results. One has to be a bit careful in generalizing.

(Oates) You have been describing these noise sources and so on, and it seems to me that this is probably with the plan of doing something about them in due course. I am wondering particularly with regard to the blades-passing noise becoming tonal noise and things of that sort. Do you foresee any possibility of reducing the manufacturing defects that are the prime source of the tonal noise? Are we at the limit already such that we can't make that better?

(Foley) One could make it better, but I don't foresee in practice it staying better very long because of practical things, like blade erosion, nicks, and things of that sort. It takes only a very slight amount of misalignment or blunting of the leading edge to start introducing these effects. I can't imagine the fan staying out of that problem in service, even if you could make a perfect fan in the plant. There are some arguments that go on to the effect that maybe one really wants combination tone because it redistributes the energy away from the primary blade-passing frequency and spreads over a spectrum. But it isn't a one-to-one correspondence at all, taking the same amount of energy and redistributing it. What you find, if you go through the calculation, is that the attenuation is different for these different waves; and so when you get out into the far field, you may find that you have more or less noise, depending upon conditions, after you distributed it into combination tones.

(Oates) As I understand this is acoustical tiling, essentially, that we would be willing to put in the inlet. Would it be better to tune that to a single, small band frequency?

(Foley) It doesn't seem to be, over this frequency range of interest. I regret that I didn't bring along the slide which I had intended to. Linings are quite effective in reducing combination tones. They don't get rid of them, but they markedly reduced the energy over quite wide ranges of frequencies. The blade-passing frequency is a little different story, because that arises not only because of the shock wave but also because of other interactions. This noise is generated by interactions that occur over the full span of the blade. Treatment at the outer walls isn't so effective on primary blade-passing frequency, apparently because it is further from the noise source.

(Lakshminarayana) Where do we stand on prediction of noise?

(Foley) The noise from the jet, of course, is in fairly good condition; but not the comparable noise prediction of the broadband noise due to the engine. One can do moderately well on the overall sound power level of the broadband noise, I think, from say a fan. Some people might object to that. But one can't do a very good job of predicting the spectrum of noise. You can only get a number for the overall level. When you have to worry about what to do with treatment, what are the effects of duct length, things of that sort, you need to know details of the spectrum. That information is mostly lacking.

(Lakshminarayana) The one thing you showed in the vortex-shedding noise; it could be the vorticity or variation of circulation, or it could be vortices shed out of the unsteady flow, or it could be the unsteady tip vortices.

(Foley) It is all left in there. I agree, we don't have it all singled out. We feel sufficiently strongly about this sort of thing that we have invested a considerable amount of money in building a new wind tunnel, specifically for the study of acoustics. We have a free-jet wind tunnel where the models can be immersed in the airstream. We surround it with an anechoic chamber, and the sound can radiate out to the far field. We have a rather substantial broadband noise program in process. There is a lot of work to be done.

(Lakshminarayana) Is tip clearance a serious problem in noise?

(Foley) Maybe somebody in General Electric would like to comment. I certainly don't have the answer to the impact of tip clearance on the noise.

(Smith) I think we will be getting some results on that type of thing under our quiet engine program being sponsored by NASA. The work is under way now, but there is still an awful lot more to do.

(Lakshminarayana) What aspects are you studying? Are you varying the tip clearance and measuring the noise level, or just the effects of this with the stator?

(Smith) We have rotors with tip clearance. We have a rotor that has tip shrouds on it, so it is not a normal type of clearance, a shroud seal clearance instead. We have removed the boundary layer from the casing, tried leading-edge treatment, and a few other things will develop as we go along. The idea is to find out, for changes in geometry that appear to be practical, what the effect on acoustic energy radiation of these changes is. It is a fairly significant program underway now.

(Hartmann) Does that study include tilted stators?

(Smith) I don't think there are any in the contract, Mel, but there is some discussion about adding that in.

(Hartmann) There are other programs with tilted stators. These as well as "feather" edge blades, clearance flows, secondary flows, etc. all probably have some effect on machinery noise. But if we are really to get the noise down to the levels desired and being discussed, we are going to have to learn to work with either lower speed machinery; or we are going to require extensive noise suppression treatments.

(Smith) I am not convinced that going to really low-speed machinery is going to help a great deal. We are testing over a range of speeds, as you know, and you certainly don't want to go to speeds that are low enough so you need more than one stage. At least it is kind of hard to conceive that that is the right thing to do, so it is a question of how low in speed you can go and still generate enough pressure to have an efficient propulsion cycle.

(Lakshminarayana) For the benefit of the people in the workshop, is there any way that you can amplify what the exact program is?

(Smith) I wonder whether, because of our limited time, it might be better not to get into that, since we do have to get to turbines.

STALL AND SURGE

Discussion Leader: Mr. J. Watkins

(Vavra) Before we go to turbines I would like to call on Mr. Watkins to introduce briefly the subject of stall and surge.

(Watkins) We sort of skipped over the subject of stall and surge earlier on; but, as you all know, Mel Hartmann has been carrying out some work on casing treatments to extend the surge line to lower flows. And we did want to hear some comments from him about this.

(Hartmann) I was recently looking at a Russian text book on turbomachinery in which they had been attempting to extend the stall margin of compressors. They had examined the effect of blockage devices and wall bleed. But I was particularly interested in the flow range over which data had been taken. They apparently send a man to the test cell and tell him to take data all the way from zero flow to choked flow. The data points were so closely spaced that it was not necessary to draw a curve between them. A good deal of data was in the stall region. This is a region that we have traditionally avoided, because we didn't think we were interested. Now we wish we had a better understanding of the stall phenomena and when we can or cannot live with stall.

Some years ago we began examining various approaches to improve compressor stall margin. Present rotor design approaches result in stall initiating in the blade tip/casing region. As flow is reduced the rotor tips load more than the other portions of the blade, and this combined with the casing boundary layer seemed to initiate the instabilities we call stall. Working on a research contract, we set out to study methods of sucking off some of the boundary layer along the casing. One approach was to place a honeycomb structure over the rotor blade tips through which flow could be extracted. The particular tests shown in Figure 1 show the design speed pressure ratio-flow curve of a rotor and the resulting stall boundary. The solid line is for the usual solid casing (no casing honeycomb), but the inlet flow was distorted by a radial screen placed upstream. The radial screen extended about 40% of the passage height from the outer wall. Consider the design speed curve which extends to the stall line. If other speed curves were indicated, they would also terminate at the stall line.

The honeycomb casing was then installed with the bleed system as indicated in the sketch. Now the design speed pressure-flow curve is shown by the dot-dashed line along with the measured stall limit line. The stall margin and flow range has been improved. Apparently removing flow from the tip region has allowed the blade loading to be increased before stall occurs. But we got a bit surprised when the bleed valve was closed. The pressure ratio-flow and stall limit lines are indicated by the dashed lines. Most of the benefit of moving the stall line resulted because of the presence of the honeycomb casing. This caused a lot of head scratching. This particular figure is with radial distortion of the inlet flow pattern. These were indicated improvements with circumferential distortions and also with uniform inlet flow.

These observations were rather exciting and led to tests of other machines with various types of casing treatments. Figure 2 shows a number of casing configurations that were run on another transonic compressor rotor. The upper left-hand sketches show a series of circumferential grooves over the rotor blading. The relative position of the rotor and the direction of rotation are also indicated. Slots cut into the casing at the blade angle are shown in the upper right-hand sketch. These were tried in both the long and short configurations shown. A honeycomb with the back closed (without plenum behind the honeycomb) is shown in the lower left-hand sketch. Skewed slots in the casing are shown in the lower right-hand sketch.

These represent some of the casing configurations that have been found to improve stall margin. The flow phenomena involved in this stall margin improvement have not been identified. We have looked at the possibility of the wall being tuned with the slots forming a series of organ pipes. The possibility that the wall could offer a different resistance to through flow and the back flow that may be associated with stall. One could think of it as a labyrinth seal that resisted back flow. Applying a capacitance that may damp the unsteadiness associated with stall has been considered. None of these phenomena seems to describe entirely what is happening in all cases.

For the time being we are stuck with some rules of thumb that seem to be significant in selecting and adjusting the casing to obtain the stall margin improvement on a particular rotor. In general we find that the mid-chordal portion of the blade must be treated to achieve stall margin improvement. We have found that only with very severe radial distortion, extending the treatment to the leading edge has been helpful. It has been noted that the region treated may require about 2/3 of the area open. This must indicate extensive fluid exchange or flow through the casing. We have not as yet been able to establish criteria for the depth of casing treatment or the volume required. One of the approaches to establish depth was to set the tube organ-pipe frequency near blade-passing frequency. Experimental studies have not indicated that this was a necessary consideration. It has become obvious that to avoid a loss in rotor efficiency that the casing treatment must not result in excessive recirculation from the rear to the front of the blade row. With these general concepts we have been able to devise casings that provide substantial stall margin improvement.

Consider the manner in which casing treatment can be applied to a fan or compressor stage. Fan stage performance is shown in Figure 3. The fan produces a pressure ratio of about 1.6 and a polytropic efficiency of about 0.86 (solid lines). The stall limit line is shown well removed from the operating line. A substantial portion of this stall margin is lost when a radial distortion is applied to the inlet flow. It can be noted by the solid curves of Figure 4 that stall limit is relatively close to the operating line. In the event that this is not adequate stall margin, the operating line must be moved to lower pressure ratio. Figure 3 would indicate that the operating line would then move to a region of lower efficiency. The dashed curves on Figures 3 and 4 indicate the performance of this fan with casing treatment. With radial distortion (Figure 4), the stall margin has been improved. Thus, moving the operating line would not be required. Figure 3 indicates that the casing treatment degraded stage efficiency only .01 in efficiency in the operating region without inlet flow distortion. This probably would be the condition for cruise. The small stall margin gain with uniform inlet is noted, but in this case probably would not be needed.

The stall margin improvement with casing treatment for four stages is shown in bar chart form in Figure 5. The 1400-ft/sec blade tip machine is the same as that shown in Figures 3 and 4. The open bar indicates the stall margin with radial distortion, and the cross-hatched bar indicates the stall margin with inlet distortion when casing treatment was applied. The improvement in stall margin with casing treatment and distorted inlet flow is such that all four stages have exhibited stall margins with distortion comparable to that obtained with uniform inlet flow.

Treating the housing in the rotor blade tip region has delayed the onset of rotating stall, increasing the stage flow range capability. However, this means that some other portion of the blading may become critical at the reduced flows. The stator hub blade elements are one such region. The sketch of Figure 6 shows grooves under the stator hub as well as casing treatment over the rotor tip. The preliminary test data did not indicate flow range improvements. The data must be examined in greater detail to determine if the stator blade elements had really become critical in this experiment.

Before closing I would like to make one more point pertaining to performance with casing treatment. All of the stages that casing treatments have been applied have been highly loaded stages. This type of stage generally indicates a substantially lower pressure ratio-flow curve when operating in the stalled region than when unstalled. The transition out of the stalled region is usually at a higher flow than the transition into stall. The hysteresis loop in highly loaded stages is generally large. It has been observed that the application of casing treatment has greatly reduced the stall hysteresis loop. This may be of great benefit in starting and staging highly loaded stages.

We hope to have some of these casing treatments in multistage compressors in the near future. I am fully convinced that there are approaches to treating casings which will allow an extension of the useful operating range of fans and compressors.

(Serovy) Does this work as well at low speeds, low PRM's?

(Hartmann) I shouldn't say it works as well. Sometimes it works better, sometimes it doesn't.

(Lakshminarayana) You mentioned something of the Russian work, what did they do?

(Hartmann) I only know what I have seen in this text book. Apparently they tried a number of schemes. They bleed flow from the outer casing. In some cases small holes to an external chamber are indicated. There appear to be some data with fences or blockage rings. In some cases grooves ahead of and behind the rotor blading are shown for bleeds. Most of the data shown indicates a rather severe efficiency penalty. But many of the casing treatments we have tried had to be trimmed or adjusted experimentally to avoid reductions in efficiency.

I do think we all should spend more time concerning ourselves with the flow phenomena associated with stalled operation.

(Watkins) There is one item of interest with which I would like to take a little of your time before handing over to turbine cooling. During testing of a transonic axial compressor, we experienced an interesting occurrence, which may be familiar to some of you.

The compressor consisted of eight stages, the first two being transonic and stages three, four, and five were quite heavily loaded with 'D' factors at the rotor hub sections of the order of .55. The compressor was to be incorporated into a turbine required to develop 1000 horsepower. As a consequence, the compressor was of small diameter (11 inches O.D.) with low tip speeds in order to retain reasonably high blade heights. The high mid-stage loadings were the result of the low tip speeds. The design was free-vortex throughout.

The test compressor was built with cantilevered stators to facilitate adjustments at this stage. The compressor hub was solid, thereby permitting no leakage to occur from the flow path. The tested compressor map was very good with a good surge line. The intermediate stages (3, 4, and 5) showed smooth stalling characteristics when plotted as pressure function versus flow coefficient. All of these stages as well as stages 1 and 2 progressed smoothly through stall at low speeds.

(McBride) About what nominal pressure coefficient?

(Watkins) I don't have this offhand. The loading factors of the middle stages were pretty high; however, for example, the 'D' factor at stage 3 hub was of the order of .55. Stages 4 and 5 values were somewhat higher than this. I re-emphasize that the stage stalling was very smooth.

The cantilevered compressor was, as stated previously, purely for test purposes, the production version being scheduled for shrouded stators. It was thought that this would be a more rugged design in the field. The shrouding was such that the seal was near the engine center line rather than at the stator inside diameter; therefore, a large cavity existed between the rotor and the stator shroud, communicative with the compressor flow path.

This shrouded-stator compressor exhibited a vastly different surge line from that of the fully cantilevered-stator compressor. The surge line literally fell apart at speeds below about 80%. If effectively coincided with the stalling line of stage 1. It appeared that stages 3, 4 and to a lesser extent stage 5 reached their stalling points, and then were unable to move smoothly over this point to lower flow coefficients. A very abrupt type of stall was exhibited by each of these stages, as distinct from the smooth type of stall with the cantilevered-stator compressor.

Had we tested the compressor first of all with the shrouded stators, we would have discarded it as being of no use for the engine application. At that time we were ignorant of the effect of external communicative cavities on stalling of compressor stages.

The two surge lines are shown on the accompanying Figure 7.

The rear stages exhibited similar pressure function/flow coefficient curves for both compressors, the only real changes being on stages 3, 4 and to a lesser extent stage 5 as previously explained.

(Unidentified) What happened at the top end?

(Watkins) At the top end, there was absolutely no change. The surge line and efficiencies were effectively the same for both compressors, as was the maximum mass flow.

It was decided to change the early stages, 1 through 4, back to cantilevered stators and to leave the remainder shrouded, this decision being based on the fact that the apparent performance changes occurred in the early and middle stages.

The compressor performance with this build hardly changed compared to that of the fully shrouded unit, i.e., the abrupt type stall of stages 3, 4, and 5 remained, and the surge line was equally bad.

We then decided to shroud the early stage stators (1 through 4) and revert to cantilevered rear stages. This resulted in almost complete recovery of the surge line to that of the fully cantilevered version. The abrupt stall disappeared on stages 3, 4, and 5.

It appeared that unshrouding the rear stages produced an upstream effect such that the critical stages in the middle of the compressor were enabled to function normally and produce progressive stalling characteristics. Shrouding the rear stages appeared to cause the upstream axial velocity profile to deteriorate in a manner to affect the radial matching of the critical middle stages.

Stage 1 had gone over stall, and the surge line was determined by stall of this stage, with stages 3, 4, and 5 operating effectively at one point (stall). At some flow and pressure ratio,stage 3 moved abruptly into stall, but at such a low pressure ratio that the surge line was hopelessly inadequate. This is a case of volumes on the inside, which apparently caused a variation in the radial gradient of axial velocity, such that the complete stage (or blade row) stalled in a different manner.

(McBride) May I add to that? There has been some speculation that the volume may be either neutral or beneficial on a machine, if it has got a hub work-flow coefficient slope that is approximately horizontal. If it is highly negative, as it tends to be in the latter stages of the machine, it is detrimental. I think this is an area that people might look into. It is a subject for some research.

(Watkins) For this particular compressor, because of the high hub loadings, the plot of temperature coefficient $\Delta T/u^2$ vs. v_A/u is quite flat.

(McBride) You are nearly constant on that all the way through.

(Watkins) Right.

(Mikolajczak) You have immediately attributed this to the volume. Have you tried filling that volume, maybe with shroud, and looking at the conclusions?

(Watkins) No, unfortunately we didn't do this. We did, however, have a similar occurrence with the fifth-stage bleed. This bleed was necessary to permit normal acceleration to occur without running into surge. The bleed flow was extracted from the casing into a plain cavity. All the time the compressor was functioning, this cavity was in communication with the compressor flow path, even with the bleed valve closed. This cavity also had a detrimental effect on the surge line. We tried filling the cavity with everything imaginable, and reduced the volume appreciably with no luck on reducing the deterioration in surge line. Once again we saw evidence of abrupt stall on the intermediate stages, but to a far less degree. But by adding this volume, which was situated across the chord of the fifth-stage stator, we worsened the surge line.

(Unidentified) Was there a flow shift at the lower speeds between the two compressors?

(Watkins) The change in maximum flow rate was erratic. For example, at 40% speed the maximum flows were the same for both the fully cantilevered and fully shrouded compressors. At 50% speed the maximum flow for the fully shrouded was larger, as it was at 70% and 80% speeds. At 70% speed the reverse occurred. The surge points at all speeds up to 80% were at a much higher flow in the case of the fully shrouded compressor, which would be the case when one considers the hang-up on stages 3, 4, and 5.

HONEYCOMB CASING TREATMENT

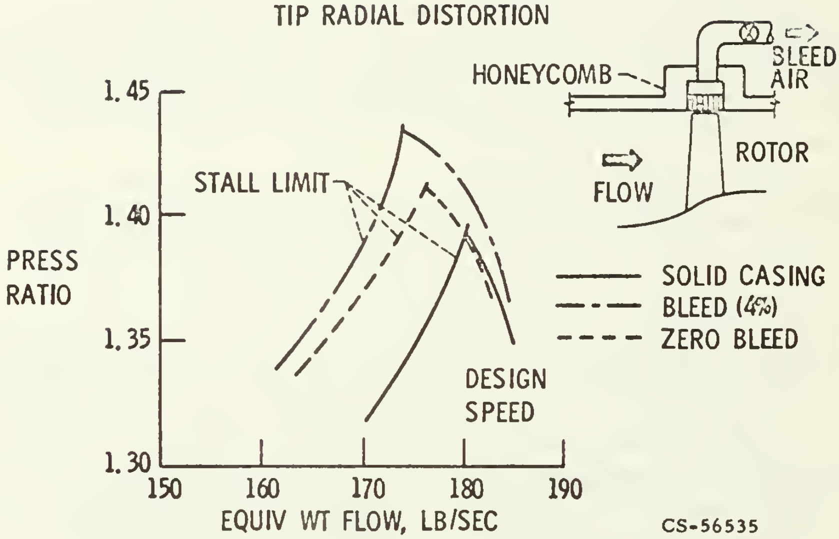


Figure 1.

VARIOUS TYPES OF CASING TREATMENT

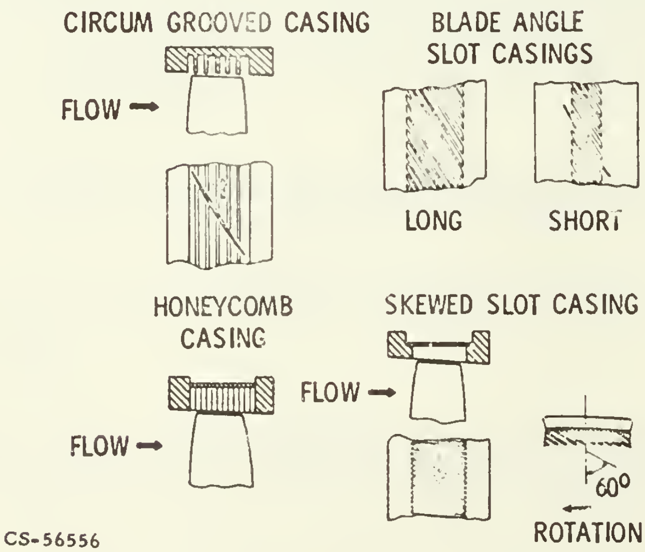


Figure 2.

STAGE PERFORMANCE WITH CASING TREATMENT AND UNIFORM FLOW

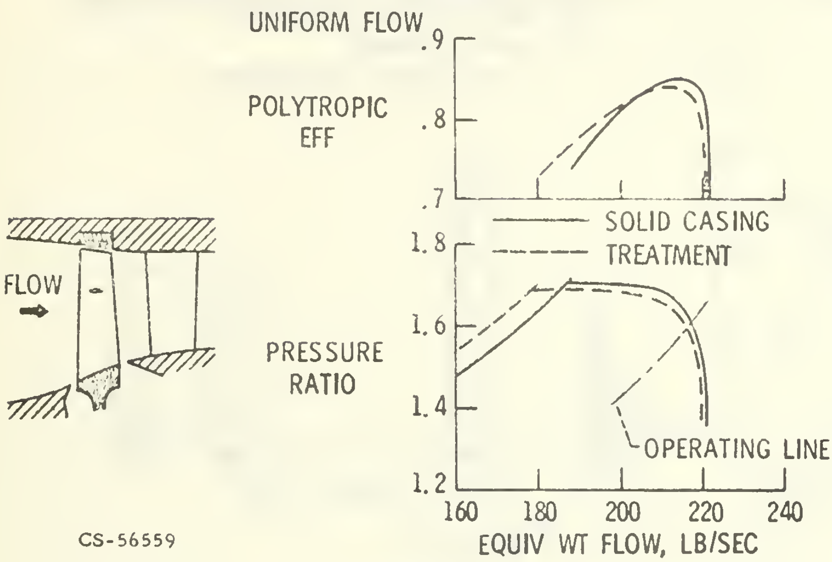


Figure 3.

STAGE PERFORMANCE WITH CASING TREATMENT AND TIP RADIAL INLET FLOW DISTORTION

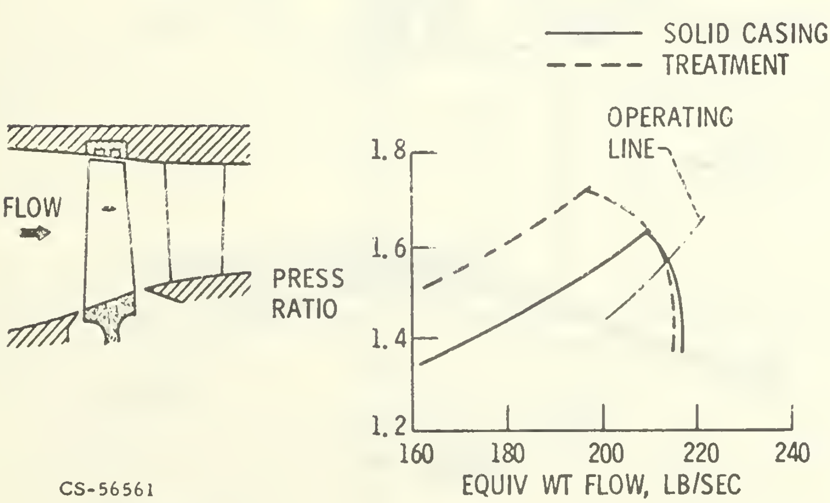


Figure 4.

EFFECT OF CASING TREATMENT
ON STALL MARGIN

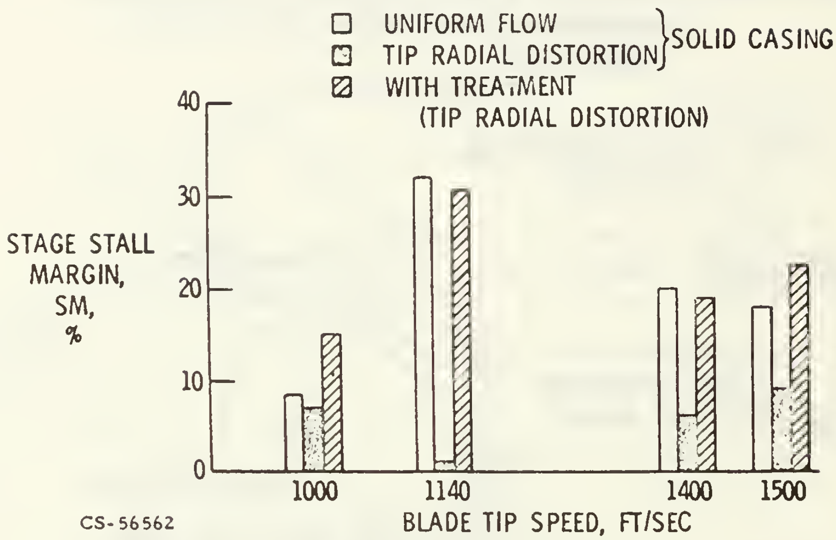


Figure 5.

ROTOR AND STATOR CASING TREATMENT

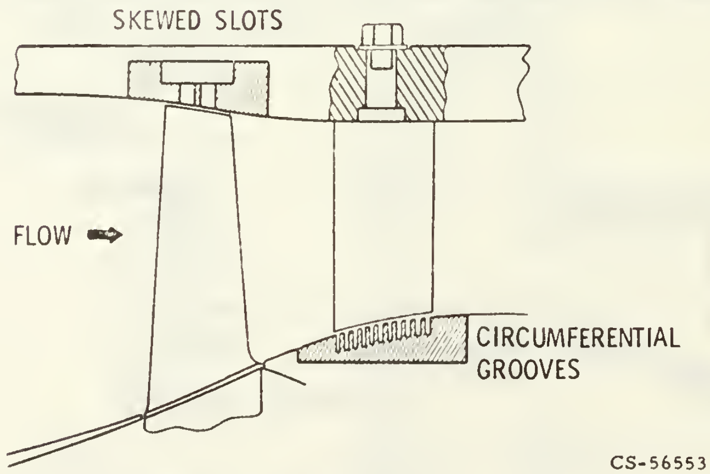


Figure 6.

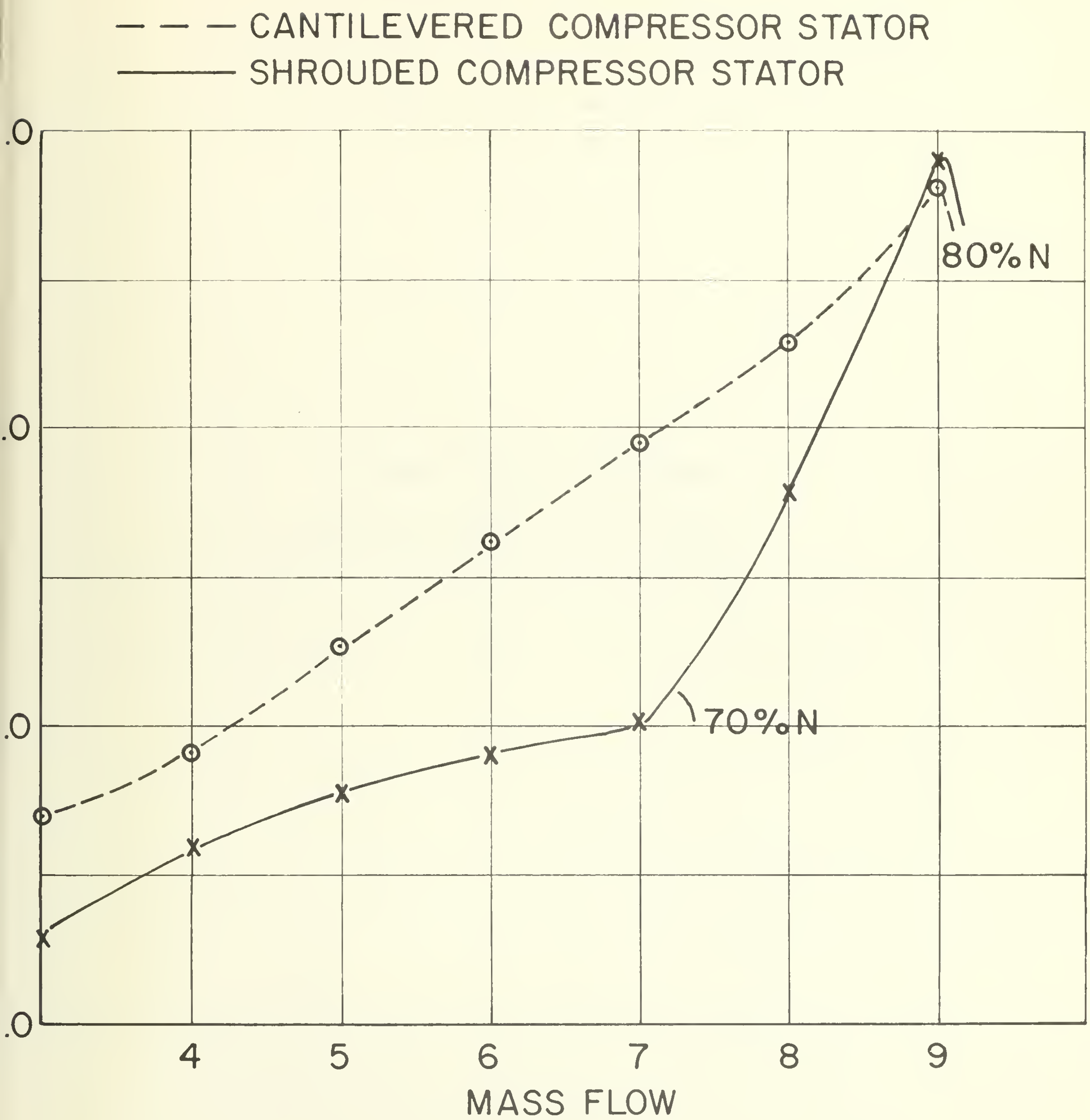

CANTILEVERED COMPRESSOR STATOR
SHROUDED COMPRESSOR STATOR
.0
.0
.0
.0
80% N
70% N
4
5
6
7
8
9
MASS FLOW

Figure 7.

TURBINE BLADE COOLING

Discussion Leader: Mr. B. E. Sells

DISCUSSION

(Sells) One of the comments that Admiral Holmquist made the other day was that he felt that we ought to address the activities of this group toward large-scale fans and turbines. So Bob Bullock and I cornered him in the cocktail session the other night to discuss this, because we had the feeling that small turbines and small compressors are more difficult to design and to achieve the predicted levels of performance and the range of operation than are the large ones. There seemed to be some misunderstanding. We didn't communicate too well, because I think the three of us went our separate ways. Two of us still believe one thing and the Admiral, who has the purse strings, still believes the other. I think this is an important consideration for this group, though, to at least go on record as saying that this is a very difficult area of design. I think it merits some consideration; because with the interest in small engines, there really is not the financial support behind these programs. Each industrial concern has to decide where it is going to place the investment dollars. Where it doesn't see production contracts, it isn't going to get strongly behind these activities. And so that effort is going to suffer until we see, I think, some strong support. I think there is some missionary work to be done in certain areas of the sponsoring organization. Twist isn't the most difficult problem. It is really end losses in these small devices, and there we understand so little about predicting them and then doing something with them. We really talked about that as one of the major elements of our large fans, turbines, and compressors. Well, so much for that.

We really decided that there wasn't time to talk much about aerodynamics of turbines and to go over this whole list. But one of the items that seems to be of great importance today is higher and higher turbine temperatures, and to this the Admiral did address himself. He even goes so far as to look forward to stoichiometric turbines. Well, one of our problems is not just running temperature, but it is running temperature for some given length of time. So really our problems then seem to resolve about temperature prediction and life prediction. And this gets us back to some other problem areas. I think we are back to boundary layer problems, although I hope we are not going to have any more discussion of that. But in reality as I see it, we are interested in the boundary layer, but not as a method of predicting losses because I think we can adequately do that. We have enough data to determine performance predictions, but we are sadly lacking in the area of temperature predictions as we go higher and higher. We have these very severe life requirements, which have gone from short life to longer and longer life; and temperature distribution then becomes very important, not just as average temperature level but purely temperature distribution around the blade surface. Houchens and I made a list of about ten or fifteen things. We could simmer that down to about two or three items that we thought we might use as teasers to get some discussion.

Since we have so little time to get some open discussion on these problems, if you would like to address to these three areas first. One is the external distribution of "h" around the blade. Then some performance effects of coolant ejection, where we have blades that possibly have film ejection at the leading edge. They have film ejection along the surface, or they have film ejection or slots at the trailing edge. All of these have problem areas in addition to the flows that may be used convectively inside and have to be reintroduced into the flow path either from, for example, nozzles or stator blades. The flow may have to be passed back into the system. I think we would like to talk a little bit about that. Then as a last thing, if we have any time, to talk about some of the problems of stoichiometry in turbines.

(McBride) Bert, going back to your first theme, why don't you mention turbine size as a parameter; because if you get too small a turbine, you can't cool it as well, and you are going to be limited in your temperature capability. In other words, you have to have a finite wall in your turbine.

(Sells) This probably reflects itself in performance, though, rather than the ability to cool. I think you can always get enough cooling air into it, because you can always make low enough aspect ratio blades and blades with thick trailing edges. The question is, can you afford to pay the performance penalty? Is there a limit to the usefulness of cooling? Particularly in the smaller sizes, do we run out of cooling capability as far as economics are concerned?

(Houchens) Bert, I would like to say a few words about the first subject, determination of the outside heat transfer coefficient. As turbine inlet temperatures go up from the current level of approximately 2500^{o}F to 3000^{o}F, the accurate calculation of outside "h" will be much more important. Now, if we miss a leading edge "h" 20%, it may mean 40 degrees in metal temperature. As the temperature difference between the gas and the metal gets much greater, accuracy of the heat transfer coefficient used in the design analysis becomes much more important.

I believe work that can be done very effectively at a university is the analytical and experimental work aimed at a more accurate determination of heat transfer coefficient on the outside of turbine blades. Work on the second item, losses due to coolant ejection, can also be done effectively here or at other good universities. Certainly the work towards improvement in boundary layer calculations is an important part of the analytical approach, but I think experimental work on heat transfer coefficient should be pursued in parallel. These experiments have to be done in the right environment. I don't believe cascade tests are the proper environment. It should be a running stage that has the high-intensity turbulence levels and other rotation effects.

(Sells) When you say a running stage, are you recommending a stage at temperature; or one that simulates conditions with warm air?

(Houchens) Well, I was thinking about rig tests, air turbine tests, ones that are a lot closer to the real conditons than cascade tests. I think that cascade tests have to be done in parallel, but you can really be misled with cascade tests only.

(Sells) I think that one of the problems that Walter is alluding to is that we, at least at General Electric, have tried to address ourselves to partially, we certainly do not really understand the problem and its solution yet, is the question of "h" at the leading edge. It is a very small area so we think this is relatively unimportant, and from a loss standpoint I think that is true. It is not significant. But from the standpoint of temperature distribution, the effect on life because of low-cycle fatigue problems, this becomes a tremendously important area for accurate predictions. The different experimenters in cascades and some of the literature that we have indicate there is as much as a 40% or 50% spread in what people think is the proper "h" that is associated with the leading edge. We have tried to do something in experimental turbine work to try to measure this effect. We are still working on it. We haven't been very successful yet. So that leads us to one of the things, the instrumentation which is needed very badly for some of these areas.

(Vavra) How do you measure this now?

(Sells) Would you like to talk to that point, Walt? Let me say first we aren't ready to publish anything on it because we are not far enough along, and we may not be right with our results so far.

(Houchens) There is an ASME paper presented by F. Burggraf at the 1970 ASME Winter Meeting which describes the techniques for the heat transfer coefficient. Essentially, little electrical heating elements are imbedded in the blade, or plate, or what have you. With a measured electrical input, measured surface temperature, and known air conditions, the heat transfer coefficient can be calculated. With proper heat loss calibration and careful correction for conduction, this method has worked well.

(Sells) When you say successful, I think the technique is successful. Adapting it into a rotating framework adds another degree of difficulty in being able to check your results. In the stationary sense you have a better check, no slip rings, etc.

(Bullock) Schlichting and Das (Flow Research on Blading, L. S. Dzung, Editor, Elsevier Publishing Company) have reported some very interesting experiments about the effects of turbulence of the order of ten percent.

(Fuhs) Some of these comments are directed to the day when TIT goes up to 500 degrees or so; but apparently even now in the area of shroud cooling with the present day TIT, there are serious problems on the extent to which you can predict heat transfer and the extent to how well you can take a given amount of air and cool the shroud region. Has that been your experience?

(Houchens) Yes, I think sometimes we worry so much about the blade that we forget about the rest of the turbine. It is going to be just as tough, and is now, to cool those bands and the shrouds and the rotor cavities, etc. So that can't be neglected.

(Fuhs) The Gas Turbine Lab at MIT is approaching this in a little different way. I don't know whether Professor Louis would want to comment on some of his work, or whether it is too early.

(Louis) It is too early.

(Dahlberg) I think in connection with our talking about secondary flows, it is appropriate to mention the effect of transport of the gas near the wall which is normally cooler down onto the airfoil, which adds to the problem of accurately predicting metal temperatures. This is a very difficult thing to handle. It is not just a two-dimensional problem on the airfoil itself, but it has got the elements of the aerodynamic problem too.

(Sells) That is a very good point, because you think in the areas of fans and compressors that even the two-dimensional cascade and the two-dimensional approach to boundary layer calculations may be satisfactory. That has been kicked back and forth. But that really doesn't satisfy this problem where cooling is concerned.

(Dahlberg) This is particularly true in small turbines. You can get a substantial cooling effect on the aft portion of the airfoil due to secondary flows that you don't normally predict.

(Sells) Bob, do you have something else that you wanted to add to this question of leading-edge temperatures?

(Bullock) K. Gersten and J. Steinheuer (Untersuchungen im Institut fur Aerodynamik auf dem Gebiet der Grenzschichtstromungen) have shown that large orders of turbulence have a vital effect on heat transfer coefficients near a stagnation point.

(Sells) There is also a Russian paper; I don't remember the author to that. It is at variance with some of the other numbers.

(Vavra) It still would be interesting to hear from Professor Louis as to what sort of ideas are behind his scheme. Is that too early too?

(Louis) I think I could describe what we have in mind. Basically we are studying the aspect of unsteadiness in the boundary layer, the casing boundary layer, particularly the possibility of instability in that boundary layer. There is a fairly large amount of potential kinetic energy. I want first to dissociate the steady flow from the unsteady flow. First I want to simulate a steady flow, the average steady flow, over the shroud, and then later on the turbine with the cooling, each time with the cooling slots. The idea is that from the heat-transfer-coefficient point of view, we want to simulate the ratio of the difference of enthalpies or temperatures. We would like to have a fairly fast response instrumentation. Therefore, that would tell us that we would like to do this at room temperature, as far as the casing temperature or the blade temperature is concerned. We would like just to cut down the temperature from the blade, which is 1600-1700 degrees Farenheit, to room temperature. So that is a factor like four. We would also like to keep the same Mach number and the same Reynolds number. This we can do. Since all the flow times, all the characteristic times, are like 100 microseconds to a millisecond, acoustic times are less than a millisecond. If the test runs for a second, that is plenty of time to do the test. That tells us we can do a blowdown. So if we scale by a factor of like four, that means the temperature of the warm gas is like 400 degrees Farenheit. The temperature of the cold gas is minus 30 or 40 degrees Farenheit, which can be cold CO_2 or liquid nitrogen. So basically what we are going to do is have a blowdown facility using a pebble-bed heater, with temperatures like 500 degrees Farenheit, and an air storage. So from the air storage, we go into the pebble-bed heater for periods like seconds, and then we go into either a static blade or the turbine blade. The turbine facility will be evacuated so that we can spin the rotor at half speed. If we take a factor of four, we know that gives the same Mach number. Then one can change the blading when we are only running at half speed. The rotor can be spinning a half speed in a vacuum, and at a given time we open a valve and start the test. Then we can measure the heat transfer rate, the pressure fluctuations, and, of course, also the rate of acceleration of the rotor to give us the performance, which is an idea similar to Professor Kerrebrock's work on the compressor. We have a similar facility for compressor work. The instrumentations are basically Kissler gages, thin-film heat-transfer gages.

(Sells) Then you avoid the equilibrium problem by just staying at room temperature?

(Louis) Yes, as far as the wall, since the amount of heat going through the wall is going to be negligible, the amount of BTU's per square inch. Since we are only going to run for periods like seconds, the walls are essentially going to be isothermal during the test. That is basically the facility for these tests. I think this could be extended to the blades. Basically we are going to try to dissociate the kind of steady effect, average flow, and then later on for the turbine try to determine what is the influence of the unsteadiness on the heat transfer as far as on the walls.

(Sells) Do you have a time table?

(Louis) Yes, we have a time table, and so far we have followed it. Within a year I hope to have some results, after about 18 months for the rotating work. Any comments? Any suggestions?

(Sells) I am a little bit puzzled as to how you are going to keep all your parameters straight. You say you are also going to do this at low density?

(Louis) No, fairly high density. I will try to keep the same Reynolds number and the same Mach number.

(Smith) I think your comment was that you evacuate the chamber and bring the turbine rotor up to speed in more or less of a vacuum.

(Louis) That is just so that we don't need a large motor to drive the turbine up to speed, so we do it in a vacuum. Then we drive it at high pressure. We have a blow-down facility which is high pressure, so that is a pressure like five or six atmospheres. The stagnation temperature will be over 400 degrees Farenheit.

(Smith) I guess you have a big flywheel attached to the rotor.

(Vavra) What are the big advantages except for the high power which you would otherwise need?

(Louis) That's right, we only need about two horsepower to drive a fairly big rotor. The air supply is like ten or twelve pounds per second, so that is a real turbine.

(Sells) What is the diameter?

(Louis) It is twenty-three inches, and is going to rotate at about 9,000 rpm.

(Sells) You would have blades then that are about two inches.

(Louis) It is a university-type facility. We don't need a large amount of power or anything like that. For heating the pebble bed, all we need is the kind of gas burner that most people have in their furnaces in Boston.

(Fuhs) In regard to heat transfer coefficient, isn't it true that in some operating regions of the turbine even when you go to higher TIT, over a sizable fraction of the chord there is going to be transitional flow; and this then once again poses a very difficult boundary layer problem on how to describe that transitional flow. We had the gentleman from United Aircraft Research Labs mention yesterday that he had a way to tackle transitional flow. I don't know if he wants to comment on that.

(Olson) All I can say is that we became aware of the problem quite a while ago, the fact that over a large part of the chord the boundary layer is in transition because of the favorable pressure gradient. This is what motivated us to develop the transitional boundary layer procedure that we have developed. All I can say is that for the cases that we have examined to date, we do get excellent predictions of the heat transfer through the transitional region. The only input into the procedure is free-stream turbulence level; and it was difficult in the test to establish what that was, so there is a bit of uncertainty there. We are in the process of trying to clear up that picture. So I think that very soon we will be able to establish the validity of the procedure and the range of applicability. I hope very soon we'll be able to publish those results.

(Fuhs) Is it necessary to put in the pressure distribution as part of the known? Is that part of your input?

(Olson) Yes, in any boundary layer procedure it is.

(Louis) Have you applied this to turbine blades, the pressure side of the blade?

(Olson) It was done to treat the boundary layer on turbine blades. For the data that we now have, it does extremely well.

(Fuhs) Do you see any reason why as you go to higher temperatures that your method might become invalid, or do you anticipate that it will remain valid at the stoichiometric?

(Olson) At the present time we have no indication that the method breaks down as you go to higher temperatures. The only thing that I can say is that as you go to the higher temperatures, you are talking not of convective cooling but of film or transpiration cooling. I think when you begin to consider these cooling schemes, you have to again examine applicability. We have no indication at the present time that the procedure won't hold up. Just on the basis of temperature level, there is no reason why it should break down.

(Dahlberg) I think it might be worth pointing out here that from an economical standpoint, the solution to these boundary layer problems is much more critical to the cooling than it is to the aerodynamics. In the development cycle of the engine, you generally prove the performance fairly early; but it then takes you many more dollars to get the durability to the point where it is a useful machine. It is much more critical, say to the mission, that the turbine doesn't come apart; whereas you might be able to stand some reduced efficiency. So I think there is really a greater incentive here to develop a useful prediction system than there is from the purely performance standpoint. They are both vital, but I mean a great deal of time is spent on getting the life of the machine in the development cycle.

(Serovy) I would just like to ask a question. Hartmann or Katsanis might be able to say something about this. A while ago I got a report that Lewis has bought a big new turbine-cooling test facility. What have you got going for these people; what do they have to offer?

(Hartmann) They are trying to get the kind of data that we've been trying to get for the last ten years. As you well know, people have been trying to make this kind of measurement and prove these kinds of things, just as we have been trying to prove all our other prediction techniques for years. We have never really had much data to compare.

(Burrows) We need to look at the effects of film and transpiration cooling on turbine performance, not only from NASA experience, but from other sources. There are indications that significant aerodynamic performance penalties may be associated with transpiration cooling. The interaction with the channel boundary layer of large quantities of coolant flow injected along the profile, end walls and between blade rows is not well understood. In the separate tests with transpiration-cooled rotor blades with turbine inlet temperatures in the order of 2,500^{o}F, stage efficiencies were 10 points less than predicted, no doubt attributable to flow separation. The first day of this meeting the benefits of stoichiometric design were advanced, but the practical feasibility of this approach has yet to be established.

(Hartmann) We need cooling methods that are not going to kill performance, or we are not going to use them.

(Fagan) I am sure you wouldn't feel free to talk about actual numbers in losses, those kinds of parameters that you know; but would it be possible for you to describe what you think the basic phenomena are, as you understand them at GE, that effect the performance? Is it the jet penetrating the boundary layer? Do you think the secondary air is incorporated in the boundary layers? What are the controlling phenomena of performance effects as you people see it?

(Sells) I guess you know what some of our configurations are. In some there is a film which is bled at the leading edge of the stator, film on the buckets. Where should this film be placed for the maximum effectiveness in cooling and the least effect on performance? As the flow first approaches the blading, you meet your first problem of how do you accomplish your goal of cooling and high performance. I guess I wouldn't like to talk numbers.

(Schwaar) I believe that before talking about the effect of cooling air injection into the main stream on the efficiency of an air-cooled turbine, we should define the efficiency in a consistent way. The definition which we use is similar to that indicated in a recent paper by P & W people. The isentropic efficiency is defined as the ratio of the effective turbine power and the sum of the powers which would be obtained by expanding isentropically and separately the main stream gas and the cooling air from their respective inlet stagnation states to the common turbine exit stagnation pressure. On that basis there are a couple of things which you can do to evaluate the effect of cooling air injection into the main stream. First, a cold air test of the actual blading with the cooling configuration, but without cooling air, yields a basic aerodynamic efficiency. A second cold test with cooling air simulation, using the correct percentage of cooling mass flow rate, would yield a correct efficiency only if, among other similarity requirements, the referred stagnation states of the main gas and cooling air would be the same as in the actual hot turbine. With cold air simulating the main gas, the cooling air should be supplied at a very low temperature, which is not the case in the test. Since the efficiency of the cooling air expansion is low, using a relatively too high cooling air temperature results in too low a turbine efficiency. This efficiency, however, can be corrected with good approximation. Assuming that the main flow expands with the same efficiency in the simulated cooling air test as in the test without cooling air simulation, an efficiency of the cooling air expansion can be determined which, applied to the expansion of the cooling air with correctly referred stagnation state conditions, yields the actual turbine efficiency value. For example, the basic aerodynamic efficiency of one of our high-spool, two-stage turbines has been cold measured at 89.6%. Cold testing with 8% cooling air flow yielded an uncorrected efficiency of 86-86.5%, and an efficiency of the cooling air expansion of 40 to 50% which, applied to the correct referred conditions, yielded a corrected efficiency of 88%.

(Sells) Does your estimate of the additional loss correlate with the actual engine performance?

(Schwaar) I cannot answer that question, because we cannot make all the measurements necessary to isolate the efficiencies of the gas generator and the power turbines in an engine. But within reasonable assumptions, the correlation with overall engine performance also is reasonable.

(Sells) I would think you would get a conservative correction, because you don't have any way to correct for the additional inefficiency.

(Schwaar) I feel that the results mentioned above constitute a good approximation. The 50% efficiency of the cooling air expansion does not appear too high, taking into account that one-third of the cooling air is re-injected downstream of the first stator and another third downstream of the first rotor; thus, a substantial portion of the cooling air works efficiently in the first and second stages.

(Sells) Do you run cascade tests also to try to correlate your losses?

(Schwar) No, not for that purpose.

(Burrows) You are saying that performance effects due to cooling flows are treated in two ways. One is the effect of the coolant ejection on the aerodynamics of the channel and the other the cycle effect. You then handle the stage as a black box in the cycle calculation.

(Schwar) That is the reason why the definition of the efficiency is an important aspect in trying to evaluate the performance.

(Burrows) With convectively cooled vanes and blades with film cooling at the trailing edge, you will not have significant aerodynamic losses due to cooling; 1 to 1.5% is probably a maximum. But for turbine inlet temperatures of 2400°F and greater where film or transpiration cooling is a necessity, I believe there will be increases in aerodynamic losses with increases in cooling air flow. Definition of the magnitude of this effect and the trade-offs involved would be an excellent subject of R&D.

(Sells) And you do have to keep books very carefully on what belongs to the cycle, and where that air is extracted, and where it is returned.

(Burrows) Advanced-technology engines are characterized by higher pressure ratios with corresponding higher cooling-air flow temperatures which compound the cooling problem.

(Herring) I would like to ask a question. Is anybody studying the fluid dynamics of combustors? If not, why not? We take the fluid dynamics through the compressor and then we pick it up again in the turbine, but what happens in the middle?

(Huffman) Experimentally we are evaluating some combustion concepts. Our work theoretically more or less parallels our overall concept of the incorporation of fluid dynamics in the engines, which is not too popular here.

(Houchens) I have a question. I believe in the boundary layer discussion you did mention transpiration, and this is something you feel can be handled in your boundary layer theories?

(Herring) Absolutely.

(Houchens) But the film cooling, ejection at discrete holes, etc., that is a little tough?

(Herring) That is questionable, yes.

(Fagan) That was the question I tried to ask. I think it would be nice to list the additional loss mechanisms we have to consider with discrete holes, etc., even though we don't get into values.

(Houchens) The open literature has been pretty skimpy on losses due to cooled turbines. There have been a few English papers and so forth. I think nearly everyone tends to calculate them by using a momentum mixing loss. This is just a pretty gross sort of thing to do, but it doesn't work too badly. If you eject the fluid, it probably does not stay in the boundary layer. I think everybody pretty much appreciates the fact that it works out beyond the boundary layer. But you can make a momentum mixing calculation, and compare it to cascade data and so forth. Perhaps it is a little crude, but it comes fairly close. I think it is on the pessimistic side. We generally tend to overcalculate the losses. When you compare them with the cascade data, you find that the losses as you have measured them weren't as great as you calculated, which may mean you didn't measure them right. But nevertheless, it seems to be a reasonable, workable way of treating the losses analytically. Then I think you always have to fall back on a series of tests, either rotating rigs or cascade tests, to really nail down the aerodynamic losses. It is an approach that needs very much to be made more sophisticated.

(Herring) I wonder if Bob Olson would be willing to comment on some of United's more detailed work in the area of discrete transpiration.

(Olson) I wouldn't like to comment on any results, only to say that we are generating what might be called design data, as well as some basic data, to develop and examine our flow models.

(Herring) Well, I was thinking more of what your flow models might be.

(Olson) I think that as far as the film-cooling case goes, it is pretty much an extension of our boundary layer work. We are using basically the same approaches, using turbulent kinetic energy equations for the turbulence model. For the discrete-hole case, it is no longer possible in the near-hole region to use the boundary layer equations. We have to go to the Navier-Stokes Equations. We are probing a little bit in that area, but I can't say that we have been successful in that area yet.

(Huffman) This work, though, should follow from some of the elliptic, full Navier-Stokes Equation solutions. I think this is the current concept.

(Olson) Yes, that is right.

(Sells) It is an area in which it is obvious that people don't want to discuss. There is too much of the proprietary aspects of it.

(Vavra) Could we maybe address ourselves to some of the fundamental investigations that should be carried out? This is really the main purpose of this assembly. We have heard what they are trying to do at MIT. For instance, what do you think could be done in an unsteady-flow wind tunnel of the type which we have here at this school? Is that a useful research tool to get an understanding of heat transfer in an unsteady flow? I was also quite surprised when Mr. Houchens mentioned the nonapplicability of cascade tunnels for obtaining meaningful results.

(Houchens) I didn't mean to imply that useful investigations couldn't be done in cascades, I just think that it needs to be confirmed with other tests.

(Vavra) Isn't that the consensus of the same opinion that was voiced earlier as far as aerodynamic performance is concerned? The wind tunnel is an interesting, cheap way of finding the approach to the problem, where you have to go. But this would all have to be confirmed by the real-life things. This is really one of the main points that came out. We have to get close to reality with our research. It was quite interesting to hear what Professor Louis is trying to do. I always say the job of the engineer is to do the job as poorly and crudely as possible and still get results. If you take all the degrees of freedom and investigate them all, you will never get there.

(Fagan) I feel that the area of coolant ejection where you have discrete holes and very localized spots certainly is of great interest, changing the boundary layers to elliptic regions. I would like to see that left on the agenda as a necessary piece of work. I am really talking about a loss model. You can establish the mechanisms of the loss model in the wind tunnel, and certainly you have to do rotor tests too. I think there is a great deal of work. Not with pure transpiration; I think we agree that boundary layer calculations work for pure transpiration. But discrete holes and slots are an entirely different question. It has not been answered yet.

(Olson) It seems to me that there could be more work done on the effect of free-stream turbulence on stagnation heat transfer. I think what is needed here is more controlled experiments, in the way of setting up the experiment and actually measuring the detailed turbulent structure approaching the blade or cylinder or whatever. I think there have been experiments done where people have put a hot wire and said they have measured the turbulence level, but it really takes a definition. You need the turbulence spectra. You need to separate temperature from velocity fluctuations. We've done some work with the hot wire downstream of a combustor and separated out temperature and velocity fluctuations, and they are of the same order of magnitude. So you can't just say turbulence level; you have to get into the detailed structure and the fluctuations.

(Foley) I guess on the basis of what Alex was talking about yesterday, the way the wakes of rotating blades interact with stationary blades, that the kind of testing that usually is proposed, where you fluctuate the overall velocity in the tunnel, is an absolutely wrong way to get the unsteady aerodynamic effect that you want. We probably need a much better model than any that I know of that anyone has used, except right in the rotating stage itself. It is a tough one to simulate with anything but the actual machine.

CLOSING REMARKS

(Vavra) Everything has to be as close to reality as possible. Then you have to try to find a way to do that with the minimum amount of money. There is no question that it will take money and it will take effort. I think we have to grow out of oversimplifying things in order to save money, because we can keep going and never get the results that we really need. These things cost time and effort. What are the most important areas of research and development? That, I think, is the important question.

(Serovy) It was mentioned earlier that we have to go on record as saying something in defense of the small gas turbine. Is it in the records sufficiently well now?

(Vavra) I think it should be on the record; I agree with you. You want to counteract the first slide that you saw from Admiral Holmquist.

(Serovy) It is not just that people want to drive around in automobiles with small gas turbines. There is this business of simulators that NASA is messing around with for one, and I happen to be mixed up with that affair. There are, of course, numerous problems that I have seen where that could well be looked into at not very high cost. They might lead to big solutions in terms of this aircraft-engine integration business that seems to be bugging all the Admirals to death. I know Langworthy up here is just looking at it. I happen to see the back of his head. I know he is very much interested in small gas turbines.

(Vavra) Mr. Smith, do you have any comments to make? In general, just what is your feeling?

(Smith) I guess I was right on the verge of making a comment right at the end of the turbine discussion. It seems to me that the point might be brought out, that in talking about applied research programs on any subject, it is important that you structure the overall program so that somewhere in the program you get the effects of all the variables that you, as an engineer, feel are going to be important to the problem. And I think in the turbine-cooling situation, certainly the first two that were listed up there are important. They are sort of the dependent variables. They are the ones that are the ultimate answer that we are trying to find, but there are an awful lot of independent variables that contribute to those. A lot of those independent variables can be evaluated in simple experiments, and evaluated with relatively simple analytical models. When this can be done, it should be done; but we shouldn't lose sight of the fact that in real-world turbines there are some pretty complicated phenomena that can only be simulated in the complete device, and that these are likely to overshadow, maybe by orders of magnitude, some of the things that we can investigate more simply. So we have to use some judgment as engineers as to what fraction of our resources we place in the simple concepts, and how much we place in the more complicated overall expensive kinds of things. So a balanced program is necessary.

(Sells) Well, I think, following along on Roy's point, he said it and it was said by a couple of other people too. This business of looking deeply into a rotating machine with some kind of flow visualization is really important, because that is the thing that helps us to understand when the simple tests don't really give us the right answer. You can use, for example, cascade testing to understand a lot of things about cooling; but you can't understand all of them, and you don't know the interrelations.

(Vavra) I think Leroy wanted to say the same thing. You should know what is really going on in the real world; and then from knowing that, you could then maybe devise relatively simple tests to put in some of the details. This is the balance that you are talking about.

(Smith) Also sometimes there are advantages to running tests that don't embody all of the phenomena that might be important. For example, as most of you know, I think low-speed testing in the full multistage environment is a very worthwhile endeavor. I know when I am doing low-speed testing that I am not getting any Mach-number effects. When I find then a certain variation of performance with the parameters that I do change, I know that these are not caused by Mach-number effects because I don't have any in the test. When I come then to the real-world compressor, where I do have Mach-number effects, it helps me to sort out the model. If I am trying to develop a high-speed compressor, I have a better feeling for what is likely to be caused by Mach number and what is likely to be caused by other phenomena, like wall boundary layer growth, leakages, and things of this kind, which you can simulate at low speeds.

To an observer who has just been sitting here listening to what has been going on over the three days, it probably sounds as though the GE people have been anti-cascade test and anti-boundary layer. That isn't particularly true. I think we do want to make the point that you have to look at the whole picture and that you can't do only the simplified studies. It doesn't mean that the simplified studies shouldn't get any attention whatsoever.

(Vavra) Like Professor Serovy said, we need these old-fashioned methods. They have to be improved. I think that is perfectly fine, if we also know what really happens in an actual machine. Engineering is an approach to problems. We don't jump in and solve it all at one time. Any more comments?

(Bullock) I have to agree with the bulk of Dr. Smith's remarks. I have seen many people do a lot of work on problems that really did not amount to a damn! It is very important to identify the real problems, and work on them. There is always a danger of taking too big a jump. For instance, we could take an immediate jump to stoichiometric temperatures. This leap would be so broad that many pertinent variables could be neglected. Moreover, short time demonstrations are misleading; if one snuffs out a candle with his fingers, he does not really prove that flesh is a good high-temperature material. If one looks at the past, he finds that progress is made when the step is big enough to make the job interesting, but not so big that the slightest slip results in catastrophe. If I had one recommendation to make to our government planning, it would be: let us reason together and decide on the size of our next step.

(Vavra) Yes, we have seen very catastrophic consequences of such actions in this country.

(McBride) Back to Bob Bullock's point, if a lot of relatively smaller steps are made rapidly to get between A and B, the energy and requirements invested are equal to the integral area of the curves. The kangaroo takes a lot of energy, whereas the flea doesn't.

(Vavra) Yes, you may be right. I think the direction in which you jump is important. I agree with the small jumps. Alex, would you want to make a few summarizing remarks?

(Mikolajczak) I agree with Roy and Bob Bullock's comments. Research on the basis of "let's work on boundary layers and hope that something comes out in the end" we cannot afford in industry or elsewhere. Usually there exists a worthwhile challenge or a problem that has to be solved. The approach has to be to select the best way of taking the steps, the rapid steps, to a solution. This may involve analysis based on simplified models, the use of cascades or incompressible rigs (we run a lot of these and find them extremely useful) or full-scale rigs. The choice has to depend on the dollars we have at our disposal and on the capability of the available personnel.

(Vavra) Mel, would you like to make a few comments?

(Hartmann) I think the idea of being selective at this time is particularly important. No matter what is being said today, we are facing a time when there will be a reduction in our available monies and resources. We are going to be faced with a shrinking staff, but we have to somehow maintain our total strength. I have seen groups that were faced with these things but made themselves stronger. This apparently is achieved by being selective in what they tackled. We have to select those methods and developments that really help us make better machinery. That is, better machinery than we are building with our present methods; and by the way, some of those are pretty good. Correlations of our present data, etc. is not enough. These will indicate limitations which we must challenge and find ways to move back or overcome. If we do not, we are not going to extend our boundaries; we are only playing in our own sandbox. We must observe and respect limitations, and then with a good plan and with good reason, try to get past them.

(Vavra) Professor Serovy, would you like to add anything?

(Serovy) I have heard a lot of good comments from Roy and Alex and Mel. I guess there is a future for the business. I am not so discouraged as I was when I came about some of the old-fashioned methods. I also think that we surely ought to encourage the boundary layer people. Have we ever been tough on them! And yet I am really fascinated by the work that Herring and all these fellows are doing, Steve Kline and all that gang. We have just got to keep needling them like this all the time and telling them what sad shape we are in, so that they go back and get concerned at least once in a while about our little difficulties.

(Hartmann) I would like to mention an approach that might be suggested by one of our admiral types. You boundary layer men design turbomachinery for one tour of duty, and you machinery men study boundary layers. It seems to me it has taken us three days to get ready to talk to each other.

(Henderson) That is not a bad statement. I would like to second that. One thing that we have learned is that if we want to deliver a good technology, we have to change the language so it sounds like old-fashioned design techniques.

(Serovy) I would like to make one more recommendation, that is that we do this again some time. I don't care whether it is here or somewhere else, but somebody had better cut loose and get a group together again to try this. We didn't talk about centrifugal compressors at all.

DISTRIBUTION LIST

1.	Defense Documentation Center Cameron Station Alexandria, Virginia 22314	20
2.	Library Code 0212 Naval Postgraduate School Monterey, California 93940	2
3.	Commanding Officer Naval Air Systems Command Navy Department Washington, D. C. 20360	1
* 4.	Rear Admiral R. J. Schneider, USN Assistant Commander for Research and Technology Naval Air Systems Command Navy Department Washington, D. C. 20360	1
* 5.	Rear Admiral C. O. Holmquist, USN Chief of Naval Research Office of Naval Research Arlington, Virginia 22218	1
* 6.	Dr. Edward S. Lamar Chief Scientist Naval Air Systems Command Navy Department Washington, D. C. 20360	1
* 7.	Dr. H. J. Mueller Research Administrator, Code 310A Naval Air Systems Command Navy Department Washington, D. C. 20360	1
* 8.	Dr. D. Abbott Professor Purdue University Lafayette, Indiana 47907	1
9.	Prof. T. C. Adamson, Jr. Aerospace Engineering Department University of Michigan Ann Arbor, Michigan 48104	1

* 10. Mr. L E. Bell, Jr. 1
Army Aviation Materiel Command
Fort Eustice, Virginia 23604

* 11. Mr. John H. Beveridge 1
Aerojet Nuclear Systems Company
Sacramento, California 95680

* 12. Mr. Robert O. Bullock 1
AiResearch Mfg. Company
Phoenix, Arizona 85034

* 13. Mr. L. T. Burrows 1
Army Aviation Materiel Command
Fort Eustice, Virginia 23604

* 14. Mr. J. E. Crouse 1
Fluid Systems Component Division
NASA-Lewis Research Center
Cleveland, Ohio 44135

* 15. Mr. Donald E. Dahlberg 1
Pratt and Whitney Aircraft
West Plam Beach, Florida 33401

* 16. Mr. James V. Davis 1
Teledyne CAE
Toledo, Ohio 43600

* 17. Dr. John R Fagan 1
Manager Theoretical Research
Detroit Diesel Allison Division
General Motors
Indianapolis, Indiana 46206

* 18. Dr. William M. Foley 1
Manager Fluid Dynamics Laboratory
United Aircraft Research Labs.
East Hartford, Connecticut 06108

* 19. Dr. Robert Goulard 1
Director, Project SQUID
Purdue University
Lafayette, Indiana 47907

* 20. LCDR R. G. Harrison, USN 1
Propulsion Division
Naval Air Systems Command
Navy Department
Washington, D. C. 20360

*21. Mr. Melvin J. Hartmann 1
Chief, Compressor Branch
NASA-Lewis Research Center
Cleveland, Ohio 44135

*22. Mr. Martin E. Hoyer 1
Supervisory Aerospace Engineer
Naval Air Propulsion Test Center
Trenton, New Jersey 08600

*23. Dr. Robert E. Henderson 1
Director of Research
Detroit Diesel Allison Division
General Motors
Indianapolis, Indiana 46206

*24. Dr. H. James Herring 1
Aero. Lab. Forrestal Campus
Princeton University
Princeton, New Jersey 08540

*25. Mr. W. B. Houchens 1
Manager, Turbine Aerodynamics and Cooling Design
General Electric Company
Cincinnati, Ohio 45215

*26. Dr. G. David Huffman 1
Principal Scientist
Detroit Diesel Allison Division
General Motors
Indianapolis, Indiana 46206

*27. Dr. Theodore Katsanis 1
NASA-Lewsi Research Center
Cleveland, Ohio 44135

*28. Dr. B. Lakshminarayana 1
Associate Professor
Pennsylvania State University
University Park, Pennsylvania 16802

*29. Mr. R. A. Langworthy 1
Army Aviation Materiel Command
Fort Eustice, Virginia 23604

*30. Prof M. L' Ecuyer 1
Project SQUID
Purdue University
Lafayette, Indiana 47907

*31. Mr. Robert W. Leko 1
Propulsion Specialist
Naval Air Development Center
Warminster, Pennsylvania 18974

*32. Major M. J. Lenzini, USMC 1
Air-to-Surface Weapons Branch
Test Operations Dept.
Naval Missile Center
Point Mugu, California 93041

*33. Dr. J. F. Louis 1
Associate Professor
MIT Gas Turbine Laboratory
Cambridge, Massachusetts 02138

*34. Mr. J. W. McBride 1
Manager, Advanced Propulsion Systems Program
General Electric Company
Cincinnati, Ohio 45215

*35. Dr. A. T. McDonald 1
Professor
Purdue University
Lafayette, Indiana 47907

*36. Dr. A. A. Mikolajczak 1
Pratt and Whitney Aircraft
East Hartford, Connecticut 06108

*37. LCDR P. A. Munroe, USN 1
COMNAVAIRPAC
San Diego, California 92112

*38. Dr. J. N. Nielson 1
Nielson Engineering and Research, Inc.
Mountain View, California 94046

*39. Prof. Gordon C. Oates 1
Dept. of Aeronautics and Astronautics
University of Washington
Seattle, Washington 98105

*40. Mr. Robert E. Olson 1
Chief, Gas Dynamics Section
United Aircraft Research Labs.
East Hartford, Connecticut 06108

*41. Mr. James R. Patton, Jr. 1
Acting Director, Power Program
Office of Naval Research
Arlington, Virginia 22218

*42. Dr. Bruce A. Reese 1
Director, Jet Propulsion Center
Purdue University
Lafayette, Indiana 47907

*43. Mr. Pierre Schwar 1
Assistant to Director of Research and Development
AVCO Lycoming Division
Stratford, Connecticut 06497

*44. Mr. B. E. Sells 1
Consulting Engineer, Aircraft Engine Group
General Electric Company
Cincinnati, Ohio 45215

*45. Dr. George K. Serovy 1
Professor
Iowa State University
Ames, Iowa 50010

*46. Dr. Fernando Sisto 1
Professor
Stevens Institute of Technology
Hoboken, New Jersey 07030

*47. Dr. Leroy H. Smith, Jr. 1
Manager, Compressor and Fan Technology Operation
General Electric Company
Cincinnati, Ohio 45215

*48. Mr. Jeffrey Watkins 1
Chief Analytical Engineer
Solar Div. of International Harvester Company
San Diego, California 92112

*49. Mr. J. S. Yampolsky 1
Gulf General Atomics Inc.
San Diego, California 92112

*50. Provost 1
Code 02
Naval Postgraduate School
Monterey, California 93940

51. Dean of Research Administration 1
Code 023
Naval Postgraduate School
Monterey, California 93940

52. Chairman 2
Department of Aeronautics
Code 57
Naval Postgraduate School
Monterey, California 93940

*53. Dr. D. J. Collins 1
Code 57Co
Naval Postgraduate School
Monterey, California 93940

*54. Dr. A. E. Fuhs 1
Code 57Fu
Naval Postgraduate School
Monterey, California 93940

*55. Dr. T. H. Gawain 1
Code 57Gn
Naval Postgraduate School
Monterey, California 93940

*56. Dr. M. F. Platzer 1
Code 57Pl
Naval Postgraduate School
Monterey, California 93940

*57. Dr. M. H. Vavra 4
Code 57Va
Naval Postgraduate School
Monterey, California 93940

*58. Dr. K. D. Papailiou 1
c/o von Karman Institute
72 Chaussee de Waterloo
Rhode-St-Genese
Belgium

59. LCDR J. R. Woods, Jr. 1
c/o Code 31
Naval Postgraduate School
Monterey, California 93940

*Workshop Participant

Unclassified

Security Classification

DOCUMENT CONTROL DATA - R & D	
Security classification of title, body of abstract and indexing annotation must be entered when the overall report is classified)	
1. ORIGINATING ACTIVITY (Corporate author) Naval Postgraduate School Monterey, California 93940	2a. REPORT SECURITY CLASSIFICATION Unclassified 2b. GROUP None
3. REPORT TITLE Proceedings of the Workshop on Flow in Turbomachines	
4. DESCRIPTIVE NOTES (Type of report and inclusive dates) Technical Report, 1971	
5. AUTHOR(S) (First name, middle initial, last name) Michael H. Vavra, Kyriacos D. Papailiou, James R. Woods, Jr.	

6. REPORT DATE	7a. TOTAL NO. OF PAGES	7b. NO. OF REFS
16 November 1971	438	184

8a. CONTRACT OR GRANT NO. b. PROJECT NO. c. AIRTASK No. A310310A/551A/1R01003010 Title (1) d.	9a. ORIGINATOR'S REPORT NUMBER(S) NPS-57VA71111A 9b. OTHER REPORT NO(S) (Any other numbers that may be assigned this report)
10. DISTRIBUTION STATEMENT Approved for public release; distribution unlimited.	
11. SUPPLEMENTARY NOTES	12. SPONSORING MILITARY ACTIVITY Naval Air Systems Command Washington, D. C. 20360

13. ABSTRACT

This report contains the proceedings of the Workshop on Flow in Turbomachines which was held in December 1970 under the sponsorship of the Naval Postgraduate School and the Naval Air Systems Command. The workshop participants included representatives from the academic community, industry, and government. The participants at the workshop are indicated by an asterisk in the distribution list.

14 KEY WORDS	LINK A		LINK B		LINK C	
	ROLE	WT	ROLE	WT	ROLE	WT
Flow in Turbomachines						
Non-Steady Phenomena						
Transonic and Supersonic Flows						
Fluid Turbulence						
Boundary Layer Calculations						
Holographic Interferometry						
Cascade Data						
Simple Radial Equilibrium						
Three-Dimensional Flow Analysis						
Axial-Flow Compressors						
Loss Evaluation Methods						
Off-Design Performance						
Flutter						
Noise						
Stall and Surge						
Turbine Blade Cooling						